

Reference Manual

DesignCAD 3D MAX

Version 12.0

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Fax: (918) 825-6359

Sales: (918) 825-4844
e-mail: sales@upperspace.com

Tech. Support: (918) 824-1159
e-mail: support@upperspace.com

Internet Home Page: <http://www.upperspace.com>

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Introduction

Thank you for purchasing DesignCAD 3D MAX. DesignCAD is a powerful but inexpensive 2-D and 3-D CAD system that puts tremendous drafting power into the hands of both professional and casual users. It has all the features professionals need, yet casual users find it easy to operate.

This manual—the *DesignCAD 3D MAX Reference Manual*—provides you with complete information about each command in DesignCAD. The information is arranged alphabetically for easy access.

System Requirements

To use DesignCAD 3D MAX, you must have, **at a minimum**, the following hardware and software installed in your computer:

- Approximately 16 to 18 megabytes of hard drive space for a compact installation.
- Microsoft Windows 95, Windows 98, Windows Me, or Windows NT (version 4.0 or later).

Although not required for running DesignCAD 3D MAX, the following hardware is recommended for better performance:

- 486DX or Pentium processor.
- at least 16 megabytes of RAM.
- SuperVGA graphics card capable of 256 or more colors.
- SuperVGA monitor capable of at least 800x600 resolution.

Typographical Conventions in the *Reference Manual*

This manual uses a few special fonts, phrases, and symbols to refer to commands and instructions.

Mouse

- When the word "click" is used alone, it means "left-click," or to press the left mouse button. When it is necessary to use the right mouse button, the manual states that explicitly. Phrases such as "click the right mouse button" and "right-click the mouse" assume that the Enable Right Click Popup Menu option in the General Options folder has been toggled off.

Keyboard

- The keys on your keyboard may not be labeled exactly as they are in this manual. All key names are shown using bold, sans serif type. For example, the "Control" key is shown as **Ctrl** and the "Enter" key is shown as **Enter**.

- Keys are sometimes used in combinations. For example, **Ctrl+F** means to hold down the **Ctrl** key while pressing the **F** key.
- "Arrow keys" is the collective name for the up arrow, down arrow, left arrow, and right arrow keys.
- To choose a command from a menu, you can use the mouse or press a key combination.

Instructions

- Specific text or numbers you must type are shown in bold, sans serif type. For example, if you are instructed to enter **13"**, you type "13.". Then press **Enter**.
- Placeholders for items that you must supply yourself, such as file names, are italicized. When the manual says to enter "**CD** *directoryname*," for example, you type "CD" followed by a space and the name of the directory.
- Menu items, settings, and various options that you are to select or use appear in non-serif type, in small capital letters. For example, "Choose the **PRINT** command from the **FILE** menu" means that you should click on the "File" menu, move the cursor down to "Print" and click again.
- At times you will be instructed to choose commands located in submenus in the Command Menu. The sequence may be indicated with a "pipe," or vertical bar. "Select **DRAW|ARCS|ARC** (CENTER, BEGIN, END)" means to select the Arc (Center, Begin, End) command, which is located in the Arcs submenu of the Draw menu.
- Unless otherwise indicated, the phrase "click the mouse" means press the left button. If another button is to be used, it is specified.
- This manual is specifically for DesignCAD 3D MAX. For simplicity, however, the program is often referred to as simply "DesignCAD."

Technical Support

If you have a question about DesignCAD 3D MAX, before you call please look in the *Reference Manual*, or the on-line Help for the solution. Remember to check the Index and Table of Contents.

If you cannot find the answer to your question in the documentation, contact the DesignCAD Technical Support Department at:

Upperspace, Corp.
Telephone: (918) 824-1159

When calling, please have the drawing in question open on screen and the *DesignCAD 3D MAX Reference Manual* at hand.

You can also send questions by electronic mail. Tech Support's e-mail address is:
support@designcad.com.

Whether you write or call, please provide the following information:

- The serial number, version name of DesignCAD (e.g., DesignCAD 3D MAX, version 12), and release date.

- The release date is located in the About DesignCAD 3D MAX dialog box. To open the About DesignCAD dialog box, select the About DesignCAD command from the Help menu.
- The type of hardware you are using.

Look for us on the World Wide Web at: <http://www.upperspace.com>.

About DesignCAD 3D MAX

DesignCAD 3D MAX is a comprehensive computer-aided design package that incorporates a full range of 2-D and 3-D drawing functions.

You can use DesignCAD to create drawings for any assignment, from simple to complex, and the finished drawing can be printed using any printer or plotter that the various 32-bit versions of Windows support.

DesignCAD 3D MAX can be customized to fit your particular application. You can create your own Custom Toolbox and even write your own DesignCAD commands using BasicCAD!

With its numerous high-end features, DesignCAD compares favorably with CAD systems costing thousands of dollars. Unlike other high-end systems though, DesignCAD is easy to learn and use. With a little practice, virtually anyone can make detailed drawings of professional quality using DesignCAD.

In addition to its 2-D Mode, DesignCAD 3D MAX is a true three-dimensional CAD system. You can use it to construct realistic 3-D models of your projects. You can show them in wireframe view, with hidden lines removed, or with full-color shading--from any viewing angle. You can also create animation files which step the viewer around your drawing in smooth increments. For example, you could start with an aerial view of a house, descend to ground level, and then walk all around it. You can even assign material properties to your creations, placing a brass doorknob on an oak door, or creating a lavatory of rose marble with chrome fittings.

If you have DesignCAD 3000, DesignCAD 2000, DesignCAD 97, or any DesignCAD 2D or DesignCAD 3D program, and DesignCAD 3D MAX, you can interchange drawings between the two applications. You can take a cross-section of a complex beam which you may have created in DesignCAD 2D, load the cross-section into DesignCAD 3D MAX and extrude it into a beam. Then you can save the extruded beam as a DesignCAD 3D MAX drawing, even with hidden lines removed! If you want to go a step further, you can extrude your floor plan into an elevation, add a roof, and save a perspective view back into DesignCAD 2D format.

DesignCAD 3D MAX imports and exports drawings in DWG, DXF and IGES formats, and also reads and writes Windows Metafiles. DesignCAD can also export WPG, RIB and WRL formats. Other imported formats include HPGL and XYZ. DesignCAD can pass drawing information to and from the Clipboard and export OLE 2.0 objects to applications that support them.

Installing DesignCAD

Using the DesignCAD Setup Program

The DesignCAD Setup program decompresses and copies the DesignCAD program and other DesignCAD files to your hard disk. The Setup program automatically creates the DesignCAD program folder and the DesignCAD 3D MAX icons.

Before you can start using DesignCAD, you must use DesignCAD Setup.

Setting up DesignCAD 3D MAX on a hard disk

Make sure that Windows is running. Before trying to install DesignCAD, make sure any other Windows applications are closed. Insert the DesignCAD CD-ROM or the first DesignCAD installation disk into the appropriate drive.

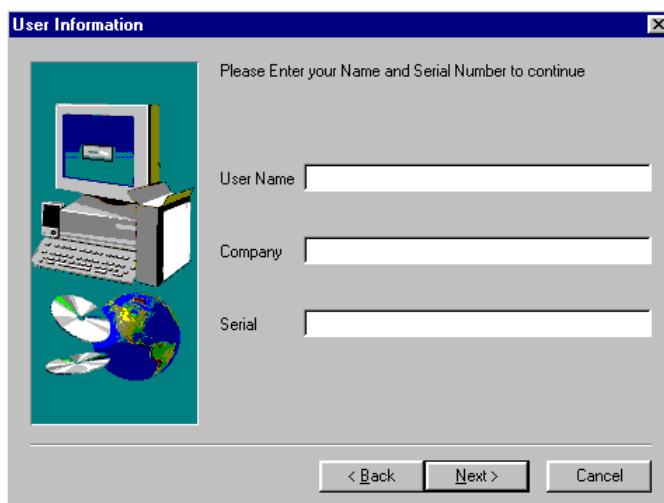
Now click the START button on the Windows Taskbar. Choose the RUN command. Windows prompts you for the appropriate drive. In the OPEN box, enter *Drive:SETUP*, where *Drive* is the floppy or CD-ROM drive you're using (a:**setup**, for example). If you need to back out of the Run command, click the CANCEL button. When you've entered the information and you are ready to move on, click the OK button.

Read the information on the Welcome screen, and then click the NEXT button.

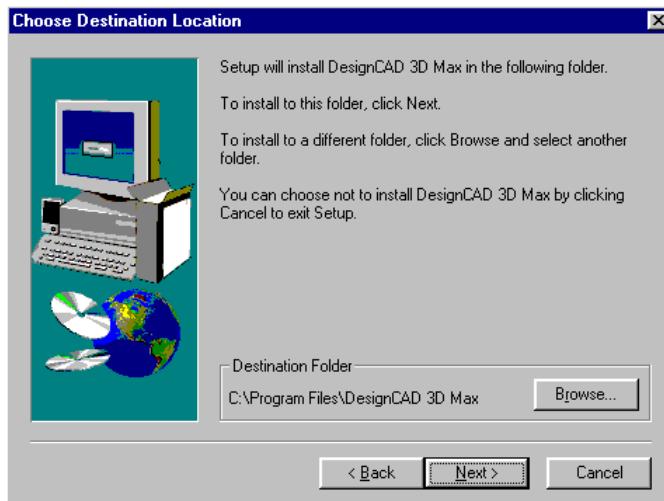


The next step is to provide registration information to the program. The name and company that Windows is registered under appear by default; change these if desired. Enter the serial number of your copy of DesignCAD. The serial number of your program is printed on a label attached to the

title page of this manual. All of the fields on this screen must have something entered in them before the Next button will be enabled. Click the NEXT button to proceed with the installation.



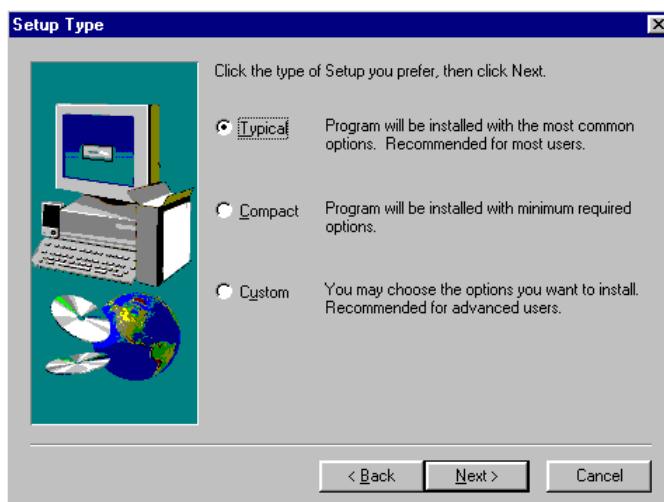
You are asked to confirm your registration information. If the information displayed is incorrect, click NO to return to the Registration dialog box and try again. If the information displayed is correct, click YES to continue. When you click YES, the Choose Destination Location dialog box appears.



To accept the suggested directory shown under DESTINATION FOLDER, click the NEXT button. To install DesignCAD in a different directory, click the BROWSE button and select the alternate location.

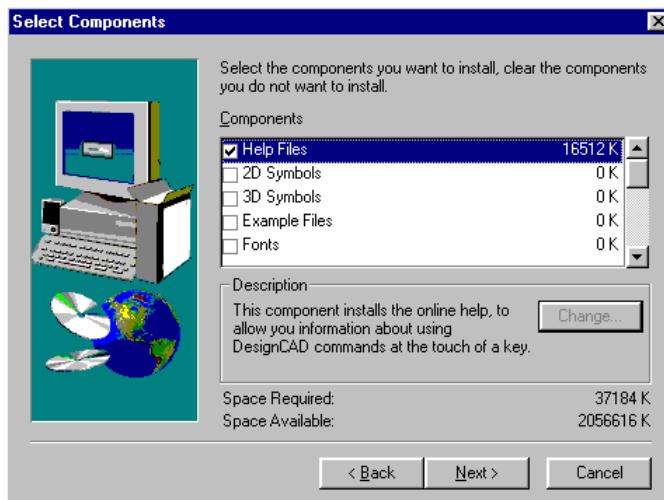


After selecting the new location, click the **OK** button to return to the previous dialog box, then click the **NEXT** button when you're ready to move on. The **Setup Type** dialog box appears.



- The **TYPICAL** installation installs all program, help, sample, symbol, and sample macro files.
- The **COMPACT** installation copies only the program and help files to your computer. However, you can later add other files—sample drawings, for example.
- The **CUSTOM** installation gives you control over which files are copied to the hard disk. You can install all or only some types of files. This is also the option to choose later if you want to reinstall files or copy files that were not copied during the original installation.

If you choose the **CUSTOM** installation, the **Select Components** box appears. To choose the components you want to install, click in the box beside the option.



Select the features you want to install and click on NEXT. The Select Program Folder appears to let you choose where the shortcuts to DesignCAD will be placed in the Windows Start menu.



Click NEXT to have the shortcuts installed to a new DesignCAD folder in the Start menu or choose a folder from the EXISTING FOLDERS area.

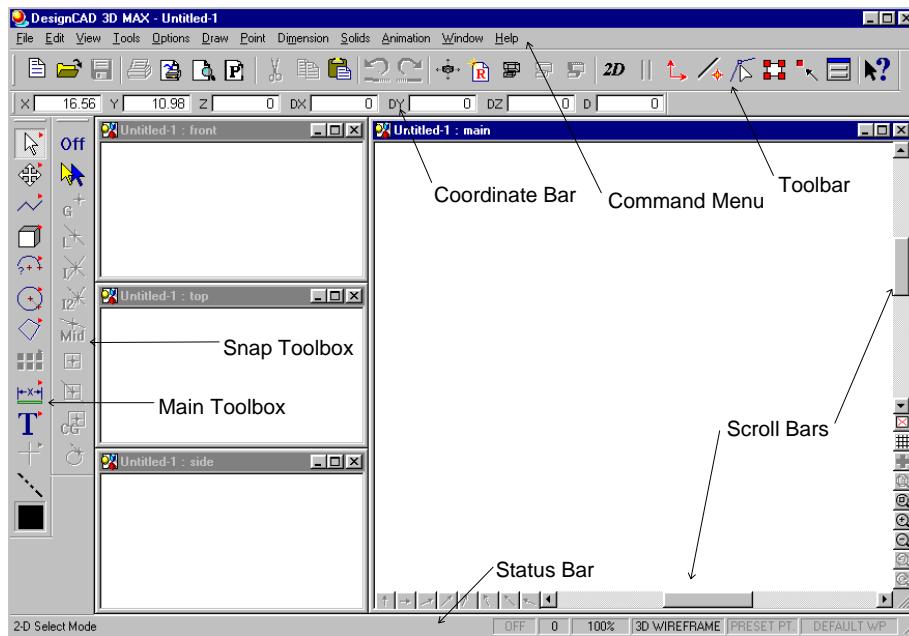
The Setup program copies the appropriate files to your hard drive. You can start the program immediately following the completion of the installation by double-clicking the DesignCAD 3D MAX icon.

Using DesignCAD 3D MAX

This section provides an overview of the drawing environment of DesignCAD 3D MAX.

The DesignCAD 3D MAX Drawing Screen

The DesignCAD 3D MAX drawing screen is shown below. Descriptions of the components follow. For details on how to maximize, minimize, open, and close windows, refer to your Windows documentation.



Color Toolbox: Changes the current drawing color or applies a specific color to selected items.



Command Menu: Contains the drop-down menu selections. From the Command Menu you can pick every available DesignCAD 3D MAX command.

Coordinate Bar: Displays the cursor's location in 2-D or 3-D space depending on whether 2-D Mode is active or not. If you are executing a drawing command, it displays the distance moved

from the last point set as DX, DY, and DZ (DZ is not displayed in 2-D Mode). The Coordinate Bar also displays the Layer list box, which shows the current layer status.

Custom Toolbox: Provides quick access to frequently used commands and macros. You can easily customize the contents of the Custom Toolbox.



Double Line Bar: Contains options for Double Line Mode.



Layer Toolbox: Contains options for viewing and hiding layers, locking and unlocking layers, and moving entities between layers.



Line Style Toolbox: Allows you to choose the current line style used by DesignCAD. Your choices affect lines, curves, arcs, circles, and ellipses. Solids and surfaces will always be drawn with solid lines of zero width. The Line Style button on the Main Toolbox acts as a toggle for displaying/hiding the Line Style Toolbox.



Main Toolbox: Contains graphic icons for the most frequently used drawing commands.



Material Toolbox: Gives you control over the color and surface texture of the items you are drawing.



Scroll Bars: Let you pan across a drawing that is larger than the screen size.

Snap Toolbox: Contains commands to set points at specific locations in the drawing, such as midpoints and intersections.



Status Bar: Displays quick help on using the current command. The Status Bar may be replaced with a progress bar for certain commands, such as the Shading command.

Title Bar: Displays the program name and the name of the currently active drawing.

Toolbar: Contains a number of icons to speed the selection of frequently used commands and functions. If the Use Single Line Dialog option in the General Options folder is enabled, the Toolbar is hidden by the Command Line when you start most drawing commands (assuming default Toolbar location). The Command Line provides options for the command you have selected and disappears when the command is completed.



View Toolbox: Controls the view settings for the currently active view window.



Accessing Commands

DesignCAD 3D MAX provides multiple ways to access commands.

Accessing Commands from the Command Menu

To access menu commands using the mouse, click on the menu title of interest. This action pulls down the list of commands available on that menu. You can now pick the command you want by clicking on it.

For the keyboard-conscious, menu commands are also available from the keyboard. You access a menu by pressing **Alt+letter**, where **letter** is the underlined letter in the menu title. To access the **FILE** menu, for example, press **Alt+F**. You then see a list of available commands, each of which also has an underlined letter. To choose a specific command when the menu is open, press the underlined letter in the command name. To use the **OPEN** command in the **FILE** menu, for example, press **Alt+F**, then **O**.

For those who like to take shortcuts, many of the commands in DesignCAD have a shortcut key. It's often much faster to go directly to a command using a shortcut rather than using the Command Menu. For example, by pressing **Ctrl+O** you can bypass the menu and immediately use the **OPEN** command.

Keyboard shortcuts are listed beside the menu commands. For example, **FIT TO WINDOW**, another command that is used frequently, has **CTRL+W** beside it on the **VIEW** menu.

Accessing Commands with the Toolbar

DesignCAD 3D MAX has a convenient Toolbar that provides push-button access to many frequently needed tasks. The available tools are shown below. To use one of these tools, click on it with the mouse.

-  Creates a new, blank drawing (New command).
-  Opens an existing user-selected drawing (Open command).
-  Saves the current drawing (Save command).
-  Bypasses the Print Dialog box and sends the current view to the printer (Fast Print command).
-  Bypasses the Print Dialog box and sends the visible portion of the current view directly to the printer (Fast Print (Current View) command).
-  Previews the way the current view would be printed (Print Preview command).
-  Sets up a multiple-view print job (Paper Space command).
-  Cuts selected entities from a drawing to the Windows Clipboard (Cut command).
-  Copies selected entities to the Windows Clipboard, leaving the original entities intact (Copy command).
-  Pastes the contents of the Windows Clipboard into the current drawing (Paste command).
-  Reverses an action performed by a command or by the user (Undo Command).
-  Reverses an action performed by Undo (Redo Command).
-  Sets the viewer position (Set View command).
-  Refreshes the screen image (Refresh command).
-  Redraws all objects on the active drawing screen (Regenerate command).
-  Provides a hidden-line view of the current view window (Hidden Line Removal command).
-  Provides a rendered (shaded) view of the current view window (Shading command).
-  This button is a toggle which determines whether DesignCAD is in 2-Mode or 3-D Mode (2-D Mode command).
-  When in 2-D Mode, this button toggles Double Line Mode on or off (Double Line Mode command).

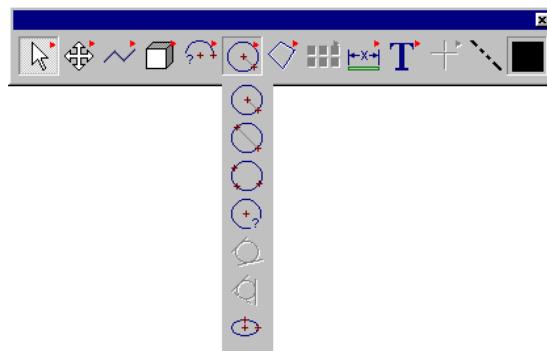
-  Toggles Orthogonal Mode on or off (Orthogonal Mode command).
-  Toggles Angle Constraint Mode on or off (Angle Constraint Mode command).
-  Toggles Point Select Mode on or off (Point Select Mode command).
-  Toggles Resizing Handles for selections on or off (Use Resizing Handles command).
-  Activates Preset Point Mode (Preset Point Mode command).
-  This button is a toggle which displays or hides the dialog box for the current drawing command (Command Dialog command).
-  To get help on something, click on this button, then click on the object in question (What's This command).

The Main Toolbox

The Main Toolbox contains tool drawers, which have push buttons for various DesignCAD drawing commands. This arrangement saves space on the drawing screen.



To see the commands stored in a drawer, click on the visible tool and continue to depress the mouse button. After a moment, the drawer slides open, revealing the tools it contains. Keeping the mouse button down, move the cursor onto the tool you want to use, and then release the mouse button. As you move the cursor over a new tool, the Status Bar at the bottom of the screen gives a brief description of that tool.

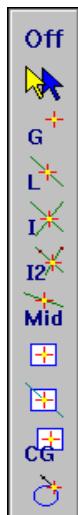


The most recently used command in a drawer is the one that shows in the toolbox. Like most of the toolboxes in DesignCAD 3D MAX, the Main Toolbox can be docked to an edge of the drawing screen or, if already docked, pulled loose to float anywhere on screen.

Clicking the Color Tool turns the Color Toolbox on or off, and shows the current drawing color. Clicking on the Line Style Tool (the button with a dashed line) opens the Line Style dialog box.

The Snap Toolbox

The Snap Toolbox (which is also dockable) contains a convenient set of tools for snapping to precise locations in your drawing. You can click on these tools even if you are in the middle of a drawing command. The **F10** key can be used to toggle the snap preview on and off.

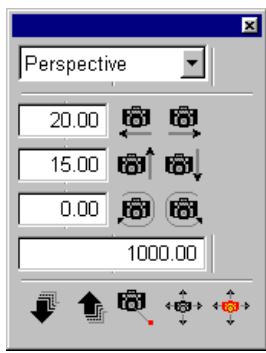


- OFF: Turns off any active snap mode.
- MOVE/SNAP: Determines whether or not the snap will set a point.
- GRAVITY POINT: Snaps to the nearest point in drawing.
- LINE SNAP: Snaps to the nearest line/curve/arc in drawing.
- INTERSECT-1: Snaps to the nearest intersection of two lines/arcs.
- INTERSECT-2: Snaps to the intersection of two chosen lines/arcs.
- MIDPOINT: Snaps to the midpoint of the selected line.
- PLANE SNAP: Snaps to the nearest point on the closest plane.
- LINE PLANE: Snaps to the intersection of a line and a plane.
- CENTER OF GRAVITY: Snaps to the center of gravity of an object.
- TANGENT SNAP: Snaps to the nearest tangent point of an entity.

The View Toolbox

The View Toolbox contains useful tools for modifying the way you look at your drawing in 3-D space. You can change the horizontal viewing angle (the viewer's rotation about the Y-axis), the vertical viewing angle (the viewer's rotation about the X-axis), and the tilt (the viewer's rotation about the Z-axis).

You can change the orientation of the drawing visually by dragging the mouse. You can choose from a set of pre-defined viewing angles, or you can define your own view. No matter what direction you want to approach your drawing from, DesignCAD's View Toolbox lets you get there. Like the snap tools, these tools can be used inside another command.



- PROJECTION LIST: Select from pre-defined settings.
- VIEWER LEFT/RIGHT: Rotates view about the Y-axis.
- VIEWER UP/DOWN: Rotates view about the X-axis.
- VIEWER TILT: Rotates the view about the Z-axis.
- INCREASE/DECREASE DISTANCE: Controls perspective.
- SET VIEWER POINTS, ROTATE VIEW BY VIEW CENTER, ROTATE VIEW BY DRAWING CENTER: These let you set specific locations for the viewer and target, or drag the view rotation manually using the mouse.

You can manually set the camera rotation angles and view distance settings. To do so, click in the numeric fields beside the buttons and type in the desired numbers. Also, the View Toolbox can be docked to the edge of the DesignCAD window.

Moving Around, Setting Points, and More

Moving Around in a 3-D Drawing

You can move the cursor in the drawing screen using either the mouse or the keyboard.

Mouse

When you use the mouse, the cursor normally moves along two of the three axes (which two depends on your viewing angles).

Make sure that the 2-D Mode button is not pushed in; this tells you that you are in 3-D Mode. If the button is pushed in, click it to turn off 2-D Mode and put DesignCAD in 3-D Mode. Next, select the OPTIONS menu, look down the menu to the 2-D Selection Mode command, and make sure there is not a check mark to the left of the command name. If there is a check mark in front of the 2-D Selection Mode command, select the command to disable 2-D Selection Mode and enable 3-D Selection Mode.

In Front View or Perspective View, the mouse moves in the X-Y plane. To move the mouse along the third axis, first choose a drawing or point command. Then simultaneously press the **Ctrl** and **Shift** keys and continue holding down the keys as you move the mouse. Forward mouse motion moves the cursor in the positive direction along the axis, and a backward motion moves the cursor in the negative direction.

When 3-D Selection Mode is used in conjunction with 3-D Mode, holding down the **Shift** key alone forces the mouse to move along one axis in the current viewing plane (X in the Front view). Pressing only **Ctrl** forces the mouse to go along the other axis in the plane (Y in the Front view).

Note: If DesignCAD is in 3-D Mode but 2-D Selection Mode, or 2-D Mode (which can only use 2-D Selection Mode), the **Ctrl** and **Shift** keys only affect the cursor while you are in a drawing command.

The mouse normally moves in increments of one screen pixel. This distance varies depending on your zoom factor and the original size of the drawing. To constrain the mouse to move in increments of a particular size (1.0, 0.25, etc.), turn on the SNAP GRID and set the Snap Grid Size by choosing GRID SETTINGS from the OPTIONS menu. If the change causes your mouse to move in a jerky manner, reduce the value in the SNAP GRID SIZE box. You may have zoomed in on a region only slightly larger than the snap increments.

Keyboard

When using the keyboard, you move about the screen using the arrow keys. The left and right arrows move you along the horizontal axis (which of the axes is "horizontal" depends on your

view angles). The up and down arrows move you along the vertical axis. Press and hold the **Shift** key while using the left, right, up, and down arrows to move along the horizontal and vertical axes in smaller increments or "steps." To move in the third direction, press the **Ctrl** key in combination with **Home** or **End**. To move the cursor in smaller increments in the third direction, press **Shift** in combination with the arrow keys or **Ctrl+Home** or **Ctrl+End**.

You can also specify the size of the cursor movement when using the arrow keys. Use the **CURSOR** command in the **OPTIONS** menu to set the **LARGE STEP SIZE** (regular arrow keys) and **SMALL STEP SIZE** (**Shift+arrow keys**) to convenient values.

Setting Points

Points form the basis of all drawing commands, determining the location of a line or curve, the diameter of a circle, or the radius of an arc.

You can select point commands from the Command Menu or the Snap Toolbox, or you can use shortcut keys. The Snap Toolbox works the same way as the Main Toolbox does. To choose a point command, click on the icon for the command.

Setting a point is easy. Press **Ins** or click the left mouse button. That's all there is to it. The right mouse button activates the Gravity command (if the **Enable Right Click Popup Menu** option in the General Options folder is disabled). Below is a table listing the point commands and their functions.

Note: When Snap commands are selected from the Snap Toolbox, you have the option of making them set a point or just move to the position; however, if you select a Snap command from the Point menu, a point will always be set.

Command	Function
GRAVITY	Snaps to the nearest point in the drawing and sets a point there.
LINE SNAP	Snaps to the nearest location on the nearest line.
INTERSECT-1	Snaps to the nearest intersection of two lines.
INTERSECT-2	Snaps to at the intersection of two chosen lines.
PLANE SNAP	Snaps to the nearest location on the nearest plane.
LINE PLANE	Snaps to the intersection of a line and a plane.
MIDPOINT	Snaps to the midpoint of the nearest line.
CENTER OF GRAVITY	Snaps to the center of gravity of the nearest entity.
TANGENT SNAP	Snaps to the nearest tangent point.
ORIGIN	Moves the origin of a drawing to a chosen point.
POINT XYZ	Sets a point using X, Y, Z coordinates.
POINT RELATIVE	Sets a point using relative X, Y, Z coordinates.
POINT POLAR	Sets a point using polar coordinates.
GRAVITY MOVE	Moves to the nearest point without setting a point.

Drawing to Scale

When you draw to scale in DesignCAD, you are usually measuring the objects in a given base unit of measure. DesignCAD doesn't care whether your base unit is meters or miles or even leagues. What's important is that you use the same base unit throughout the drawing.

Let's say that you're drawing a house and the front wall is 32 feet long. To DesignCAD, it is 32 Drawing Units. You can draw a line by choosing the Line command, setting the first point (click the mouse or press **Ins**), then specifying that the next point is 32 Units away.

But what if the next item you measure in the same drawing is 10 centimeters tall? If you draw it at a height of 10 units, it will be much too large. Why? Because centimeters and feet are different units. In this example, you would need to convert the centimeters to feet, and then tell DesignCAD the size of the item in feet.

The key point is not to mix units. If feet are convenient, call out all distances in the drawing in feet. If centimeters are convenient, measure everything in centimeters. As long as you're consistent, all is well. If you use feet, many of the commands allow you to enter distances in feet and inches: for example, to specify a line 9 feet 5 inches long, set the first point, then enter **9'5"** in the DX field of the Point Relative command.

Printing to Scale

What about printing to scale? The Print screen shows the paper units selected and the scale of the printout. Scale here represents the number of paper units it takes to print one drawing unit.

Suppose we have drawn a 10-inch box, which we specified as 10 units when we drew it. Now we want to print it at 0.25 scale. The paper units default to inches, and our drawing is also in inches. Therefore it takes 0.25 inches on paper to represent one inch in the drawing. Our 10-inch box comes out 2.5 inches long on paper.

If our box had been specified at 10 feet instead of 10 inches, then it would take *1/4 inch* of paper to represent *1 foot* in the drawing. This is a real-world scale of *1/48*, or *0.0208333*. But to DesignCAD units are just units, so the scale is still shown as 0.25. It took *0.25 paper units* (inches) to represent one *drawing unit* (feet). Printing scale is the length on paper that will represent one drawing unit.

We can show it as an equation:

$$\text{True Scale} = \text{Scale/Ratio} \text{ (where Ratio is the number of paper units in one drawing unit)}$$

In the example above, our scale is *.25*. The paper unit is in inches, the drawing unit in feet. Since there are twelve inches in one foot, the Ratio equals twelve.

$$\text{True Scale} = \text{Scale/Ratio} = .25/12 = 1/48$$

What if your paper is too small to print at the scale you need? No problem. DesignCAD can print out your drawing in panels, all to scale, which you can then assemble into a composite drawing.

Here is a list of common Architectural and Engineering scales.

Architectural

Scale Name	Actual Decimal Value
1/16" = 1'	.0625
3/32" = 1'	.09375
1/8" = 1'	.125
3/16" = 1'	.1875
1/4" = 1'	.25
3/8" = 1'	.375
1/2" = 1'	.5
3/4" = 1'	.75
1" = 1'	1.0
1.5" = 1'	1.5
3" = 1'	3.0

Engineering

Scale Name	Actual Decimal Value
1:1000 1000	.001
1:500 500	.002
1:333 333	.003
1:100 100	.01
1:50 50	.02
1:20 20	.05
1:10 10	.1
1:5 5	.2
1:3 3	.33
1:2 2	.5
2:1 .5	2
3:1 .33	3
5:1 .2	5

Selecting Objects

Many of DesignCAD's commands work only on selected objects. Other commands work on the entire drawing or selected items only, depending on the options you choose. Below are instructions for selection tasks you'll often use in your drawings.

2-D Selection Mode

In 2-D Selection Mode the cursor assumes an arrow shape. To change to this mode, press **Ctrl+2** or choose **OPTIONS** in the Command Menu and then **2-D SELECTION MODE**. When you are in 2-D

Selection Mode, you can always select an object, either by clicking it or snapping to it with point commands. In 3-D Selection Mode that is not always the case.

3-D Selection Mode

The 3-D cursor is formed by three intersecting lines—a black one along the X-axis, a red one on the Y-axis, and a green one on the Z-axis. Press **Ctrl+3** to switch to 3-D Selection Mode, or choose **OPTIONS** in the Command Menu and then pick **2-D SELECTION MODE** to uncheck it. 3-D Selection Mode only works in 3-D Mode (i.e., the **2-D Mode** command in the Options menu cannot be checked).

One point to remember about 3-D Selection Mode is that the cursor moves in 3-D space. Sometimes, therefore, the cursor looks as if it is on or near an object you're trying to select, but it is actually far from the object. For this reason, point snap commands and visual point placement may appear not to work very well in 3-D Selection Mode. However, if you check the other views, you'll see why you missed your intended target—you just *thought* you were close enough!

Note: Use 2-D Selection Mode for all the drawings in this manual. After you start a new drawing or open an existing one, press **Ctrl+2** or use the Command Menu to change to 2-D Selection Mode. In 2-D Selection Mode, the cursor is a wide arrow when there is not a drawing or editing command active.

Selecting a single object

Move the arrow-shaped cursor near the object and click. If you are close enough, the object is selected, turning magenta and showing a blue bull's-eye where you clicked. The bull's-eye is the selection handle. If you click too far away, either nothing gets selected or something closer to the cursor than the object you want gets selected. Pressing **Esc** will clear the current selection, as will clicking in a blank region of the screen.

Note: Pressing **Delete** **erases** the current selection.

With the Gravity command, it's easy to select an object and simultaneously set the handle at an exact location in the object. To set a **GRAVITY** point, right-click the mouse (if the **Enable Right Click Popup Menu** option in the **General Options** folder is disabled) or press the **.** (period) key with the cursor near the desired point in the object you want to select. The cursor snaps to the point placing the selection handle there and selecting the composite object that contains that point.

Selecting a group of objects in a region

Move the mouse to one corner of the region. Hold down the left button and drag a selection rectangle around the region. Release the mouse button at the opposite corner. Every item that was completely enclosed in the region is now selected. The handle is placed at the center of the area that the selection rectangle encompassed.

Selecting objects enclosed in or touching a selection rectangle

Click and drag the selection rectangle, but press **Ctrl** before releasing the mouse button. (Remember not to hold down **Ctrl** as you drag; if you do, you'll restrict the mouse's movement to

only one direction.) All objects that were enclosed by or touching the selection rectangle when you released the mouse button are selected.

Adding or deselecting a single item

To add a single item to the selection set, or to deselect a single item in a selection set, move the mouse near the item, press **Shift**, and click the left mouse button.

Adding a group of objects to the selection set

After selecting one group of objects, drag a selection rectangle around another group as in normal selection, but press **Shift** before releasing the mouse button. The items must be completely enclosed by the rectangle to be affected. Any items in the region which were already selected will be de-selected. (If you are in 3-D Mode and have 3-D Selection Mode turned on, remember not to hold the **Shift** key down as you drag, or you'll force the mouse to move along a single axis.)

Selecting specific items

Sometimes you need to be even more specific about which items you want to select. If DesignCAD is in 3-D Mode, you can select objects by dragging a 3-D rectangle around them. If you press **Ctrl+3** while your arrow cursor is showing, it turns into a 3-D cursor like the one you see when you draw a line. This signifies that you are now in 3-D Selection Mode. You select items as before, but for regional selections, you must enclose the items in a 3-D selection box rather than a simple 2-D rectangle. To return to the 2-D Selection Mode, press **Ctrl+2**.

Selecting all objects

Finally, let's not overlook the convenience of the **SELECT ALL** command. This command, located in the **EDIT** menu, selects everything in the drawing.

Using Selection Handles

When you work with selected objects, you often need some way to specify how DesignCAD will manipulate the items. This is the purpose of selection handles. The selection handles are the specific points in the object with or around which DesignCAD moves, copies, and rotates the selected object or objects. A moved object or copy of an object is built around the handles according to the same relative relationships as the original object.

In many cases, you just want to move your object, or a copy of it, somewhere else in the drawing. For these tasks, one handle is enough.

At other times, however, you might need to control the size and orientation of the moved or copied object. Now you need two, or possibly even three handles by which to locate the object. The primary handle (Handle 0 if you just selected the object, the blue bull's-eye if you have used the **Set Handles** command) establishes the starting location for the object. Handle 1 provides a reference point for both the direction and scale of the object's primary axis. Handle 2 provides a second directional reference for orientation.

Placing specific handles

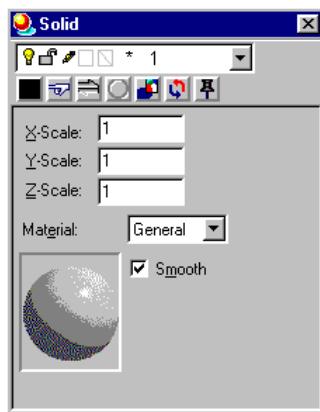
You can set specific selection handles on a selected object or group of objects by pressing **Ctrl+H** or using **EDIT | SELECTION EDIT | SET HANDLES**. The Status Bar prompts you to set one, two, or three points for the handles. If you set less than three, press **Enter** to end the command.

You can use any of the point commands to set these handles, including **Point XYZ**, **Point Relative**, and **Gravity**. You can set or change your handles even if you have already started another command.

More Information—The Info Box.

Suppose you have selected an item, but you don't know what its properties are. The Info Box provides you with the answers. It tells you what kind of drawing object the item is, what layer the object is in, the object's color, any texture mapping information, and its material.

The Info Box also gives you the ability to change those object properties directly.



Using the Info Box

Let's say we have selected a 3-D box, which is a solid. Suppose we want to double its size, expanding it from the top downward. To do this, first select the box and place a handle anywhere along its top face. Next, press **Ctrl+I**, or pick **VIEW | INFO BOX**, to show the Info Box. Change the **Y-SCALE** to **2.0** and click or **Tab** into a different field. The box grows downward to twice its original height.

If we want it to grow upward from the bottom, we would place the handle on the bottom edge. Similarly, we could transform this box into a slab of marble by using the **MATERIAL** list box in the Info Box to select a marble texture.

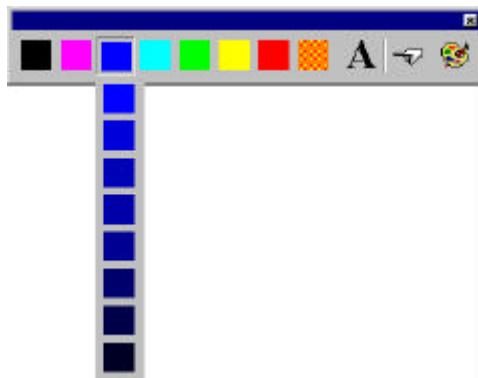
All This and Color, Too...

DesignCAD 3D MAX gives you 64 basic drawing colors. By default there are seven base colors in eight deepening shades each, and eight other lively colors. However, if the supplied colors don't meet your needs or fit your tastes, you can create your own.

The Color Toolbox has eight color buttons, an "A" button, and a "pointing finger" button. The A (for "Apply") button applies the current drawing color to the items you have selected. The

pointing finger is used to grab the color of the pointed-to object and make that the current drawing color. The other eight buttons are available drawing colors.

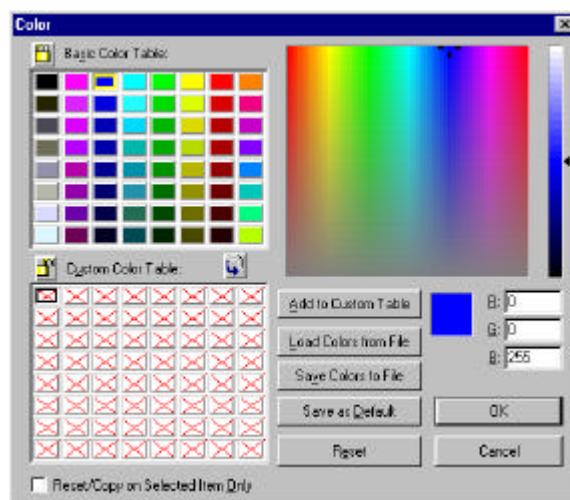
How do you get to all 64 colors? Click on one of the color buttons and hold down the button. This opens the color drawer, which shows eight colors hidden under one tool. These work like the tools in the Main Toolbox: click, hold down, slide the pointer over to the color you want, and release. The color you picked shows up at the bottom of the Main Toolbox, indicating that it is now the active drawing color. It also becomes the top color in its drawer.



Editing a Color

If you want to edit a color, make that color the top one in its drawer. Then double-click on it to activate the CUSTOM COLOR command (this command is also available through the Options menu).

Color dialog box



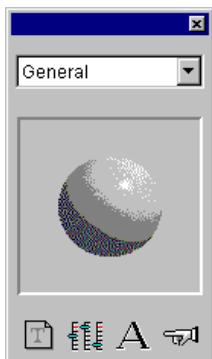
Choose CUSTOM COLOR command. Move the Color Selector (the reticle in the box with all the colors) to change the current color in the Color | Solid box. Moving left or right will change the

hue of the color and moving up or down will change the saturation of the color. The brightness of the color can be adjusted by moving the arrow beside the brightness bar. A custom color can also be set by entering new values in the Red, Blue, and Green boxes.

If you want to use the color later in the same drawing, click the ADD TO CUSTOM TABLE button, then click **OK** or press **Enter**. If you just want to use the color for the current drawing color and do not wish to add it to the list of custom colors, just click **OK** or press **Enter**. The custom color appears in the Main Toolbox as the current color, and new objects will be drawn in that color.

The Material Toolbox

DesignCAD 3D MAX comes with more than 20 pre-defined material types for you to draw with. Using the Material Toolbox allows creation of new materials from a set of customizable textures and properties. Any material can be applied to any object in a drawing.



The Material Toolbox has seven areas of interest:

1. The control button at the upper-right corner
2. The Current Material list box
3. The Material Preview box
4. The New Material tool
5. The Edit Material tool
6. The Apply Material to Selection tool
7. The Same Material As tool

You can dock the Material Toolbox to the side of the screen, let it float, or turn it off. The control button (the "x" sign in the upper-right corner) is used to turn the toolbox off. Use the blue bar beside it to drag the toolbox around the screen.

If you click on the down arrow, the Material box shows a list of material types available for use. The list includes any materials you have created and added to the list.

Note: Don't confuse this list with the Material List command in the File menu. The Material List command provides a net listing of all the attributes you placed in your drawing. Material, as it is referred to in the Material Toolbox, affects the appearance of shaded drawings. Attributes are generally used to denote physical materials—lumber, bolts, screws, and other parts—used to build the object depicted in your drawing.

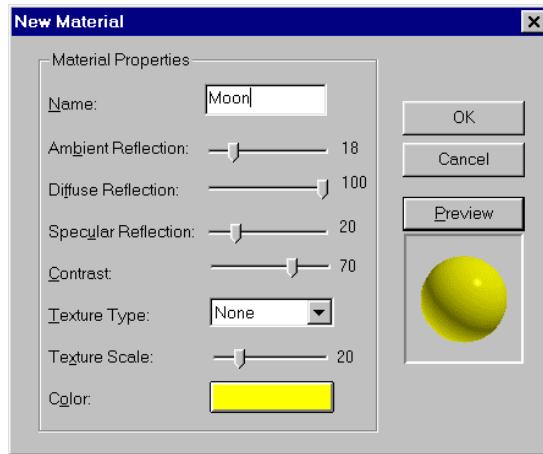


The Material Preview box shows how a sphere of that material looks when shaded.



The New Material tool opens up a miniature workshop where you design your own new

substances for DesignCAD to render:

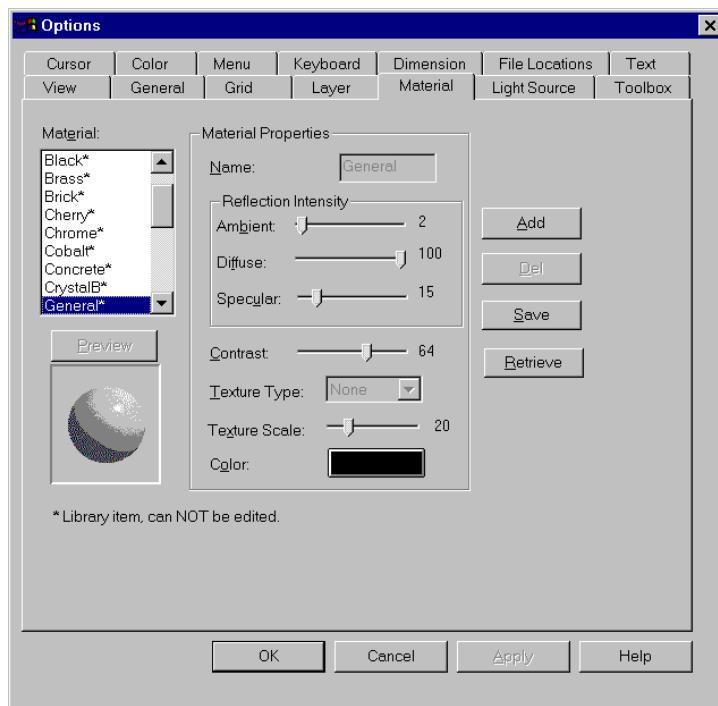


The example above shows the new (created) material MOON, a pale, rough substance.

Click CANCEL to discard your changes and return to the drawing. Choose the OK button to accept the current settings; the Material Options folder opens to show that the new material has been added to the list of available materials. Click OK to return to the drawing.



The Edit Material button brings up almost the same screen, with additional buttons for Add, Delete, Save, and Retrieve.



From this screen you can edit the current material (standard library materials cannot be edited). You can also save your new material to disk, retrieve a material from disk for further editing, delete a material from the list, or add another new material. If you choose Add, the current material properties become the default settings in the New Material screen. You can edit them there.

In editing properties, the higher the number, the more noticeable that property is in your drawing.

Material Properties:

Name: the name of the new substance (limited to 16 characters or less).

Ambient: the amount of random background light reflected on the object.

Diffuse: controls the percentage of light the object reflects.

Specular: controls how shiny the object is.

Contrast: controls the degree of difference between light and dark areas.

Texture Type: the kind of surface the object has: none, sandy, marbleized, etc.

Texture Scale: controls the coarseness of the texture.

Color: sets the default color of the material.

DesignCAD 3D MAX Command Reference

The "Command Reference" section lists in alphabetical order the commands available in DesignCAD 3D MAX. Each listing gives information on how to select and use the command. Examples are provided for many of the commands.

2-D Mode Command

Menu:	OPTIONS
Menu Command:	2-D MODE
Toolbar Icon:	

The 2-D Mode command shifts the viewer perspective to a front view and treats operations as if they were only occurring in 2-D space. This makes it very easy to use DesignCAD 3D MAX to draw just as you would in a regular 2-D drafting package.

If your drawing contains 3-D objects, it will still be a 3-D drawing; however the cursor will only move parallel to the X and Y axes, with the Z-coordinate set to zero. This makes it very easy to add 2-D information to your drawing.

Using the command

Choose 2-D MODE from the OPTIONS menu. The command acts as a toggle. When you are in 2-D Mode, the menu will have a check beside the command. Choosing the command will toggle the drawing mode into or out of 2-D mode depending on the current mode.

Note: In 2-D Mode, the Trim commands ignore the 3-D aspects of any existing lines, and treat them as flat projections onto the XY plane. This means you can trim lines that would never meet in 3-D space against each other's Front-View projections in 2-D space. This is a powerful feature, but you must pay careful attention to what you are doing.

2-D Selection Mode

Menu:	OPTIONS
Menu Command:	2-D SELECTION MODE
Shortcut Key:	Ctrl+2
Toolbox Icon:	
Point 1:	Object to be selected or first corner of selection rectangle
Point 2:	Second corner of selection rectangle (optional)

Most DesignCAD 3D MAX commands require drawing objects to be selected. For example, to erase an object you can select it and then press the **Del** key.

The 2-D Selection Mode causes the selections to take place with respect to the drawing screen. When you click on an object, the object closest to the cursor on the screen will be selected, not the closest object in 3-D space. The two are not always the same.

Using the Command

There are four ways to select objects in DesignCAD 3D MAX:

- Click directly on the object.
- Drag a selection rectangle around a region.
- Choose the **SELECT ALL** command.
- Choose the **SELECT ENTITY** command. This method only works while the program is in Preset Point Mode.

If you drag a selection rectangle around a region, objects inside that rectangle will be included in the selection. To include objects that touch the selection rectangle, hold the **Ctrl** key down during the selection.

The **Shift** key can be used to add items to the current selection set. For example, to select two objects in the drawing, you can select one and then hold the **Shift** key down while you select the other.

When you select something by dragging a selection rectangle in 2-D Selection Mode, the objects inside the rectangle are selected, regardless of their “depth.” With 3-D Selection Mode you drag a three-dimensional box, and objects must lie inside the box in all three directions to be selected.

Note: A DesignCAD drawing is always in either 2-D Selection Mode or 3-D Selection Mode. To change the selection mode, select the **2-D SELECTION MODE** command from the **OPTIONS** menu which toggles DesignCAD into 2-D Selection Mode (and out of 3-D Selection Mode) or vice versa.

You may also select the 2-D Selection Mode tool or the 3-D Selection Mode tool in the Main Toolbox. If 3-D Selection Mode is active, the 3-D Selection Mode tool will be visible in the Main Toolbox. Click and hold on the **3-D SELECTION MODE** tool; the tool drawer will slide out. While still holding the mouse button, move the cursor over the **2-D SELECTION MODE** tool. Release the mouse button to put the currently active drawing in 2-D Selection Mode.

In 2-D Selection Mode, the cursor is the familiar mouse arrow. In 3-D Selection Mode, the cursor is a 3-D cursor made up of three lines parallel to the X, Y, and Z axes.

Example: Draw any object on the screen and select it.

Move the cursor to any point on the object and click the left mouse button. The object changes color to signify that it is selected.

See Also: **3-D Selection Mode**

3-D Selection Mode

Shortcut Key: **Ctrl+3**



Toolbox Icon:

Point 1: One corner of the selection box

Point 2: Opposite corner of the selection box

The 3-D Selection Mode changes the selection mode of DesignCAD 3D MAX so that selections are made in 3-D space rather than with respect to the drawing screen.

In 3-D Selection Mode, the selection takes place in three dimensions. When you select an object by clicking on it, the cursor must be on the object on the X, Y, and the Z axis. When you select an object by dragging a box around it, the object must be enclosed along X, Y, and Z axes, not just the left, right, top, and bottom.

Using the Command

There are three ways to select objects in DesignCAD 3D MAX:

- Click directly on the object.
- Drag a selection rectangle around a region.
- Choose the **SELECT ALL** command.

If you drag a selection rectangle around a region, objects inside that rectangle will be included in the selection. To include objects that touch the selection rectangle, hold the **Ctrl** key down just before you release the mouse button as you complete the selection rectangle. Don't hold it down while dragging the cursor.

The **Shift** key can be used to add items to the current selection set. For example, to select two objects in the drawing, you can select one and then hold the **Shift** key down while you select the other.

When using commands like Gravity in 3-D Selection Mode, the cursor snaps to the point that is closest in 3-D space, not necessarily the one that appears to be nearest the cursor on the screen. It's easy to use 3-D Selection Mode using the DesignCAD Tile views. This allows you to see objects from all sides.

Note: A DesignCAD drawing is always in either 2-D Selection Mode or 3-D Selection Mode. To change the selection mode, select the **2-D SELECTION MODE** command from the **OPTIONS** menu which toggles DesignCAD into 2-D Selection Mode (and out of 3-D Selection Mode) or vice versa.

You may also select the 2-D Selection Mode tool or the 3-D Selection Mode tool in the Main Toolbox. If 2-D Selection Mode is active, the 2-D Selection Mode tool will be visible in the Main Toolbox. Click and hold on the 2-D SELECTION MODE tool; the tool drawer will slide out. While still holding the mouse button, move the cursor over the 3-D SELECTION MODE tool. Release the mouse button to put the currently active drawing in 3-D Selection Mode.

In 3-D Selection Mode, the cursor is a 3-D cursor made up of three lines parallel to the X, Y, and Z axes. In 2-D Selection Mode, the cursor is the familiar mouse arrow.

Example: Select an object using a selection box.

Make sure that your view windows are in the DesignCAD Tile default arrangement. (See the "DesignCAD Tile Command" entry for details.) Draw an object on the screen. Next, choose 3-D SELECTION MODE and select the object with a 3-D selection box. Make sure the object is completely inside the selection box along all three axes before setting the second point.

Hint: To ensure that you are enclosing the entire object in all 3 dimensions, choose the DesignCAD Tile command to set up your main Perspective view and smaller Front, Top, and Side views. Move the cursor so that it appears outside the "bottom-left" corner of the object to be selected in all three of the smaller views. (You may have to press **Ctrl+Shift** while moving the mouse to get there). Now press the left mouse button, hold it down, and move so that the cursor drags the selection box past the "top-right" corner of the object in all three of the smaller views. Release the mouse button. If the selection box completely enclosed the object, it will be selected. (It takes some practice, so keep trying.)

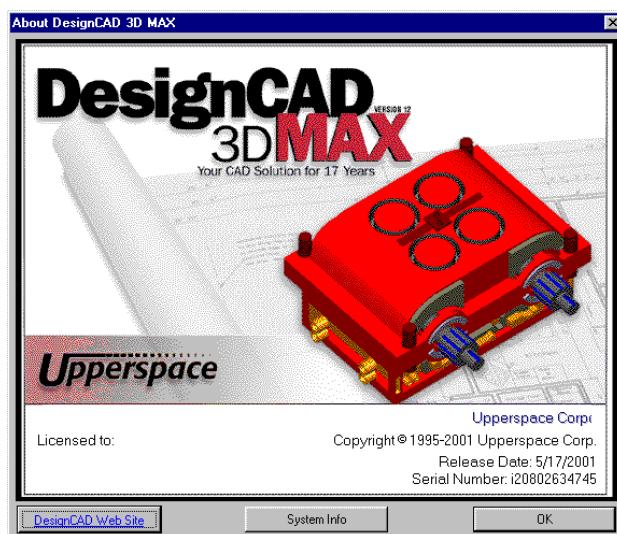
See Also: 2-D Selection Mode**About DesignCAD Command**

Menu:	HELP
Menu Command:	ABOUT DESIGNCAD

The About DesignCAD command displays information about the program, including the DesignCAD release date, who the program is registered to, and the serial number. The command also offers a direct Internet links the DesignCAD home page on the World Wide Web. You can also find information about your computer system.

Using the Command

Select the ABOUT DESIGNCAD command from the HELP menu. The About DesignCAD dialog box appears.



Using Internet links

If you have a network or dial-up connection to the Internet and Web browser software installed, you can go directly to the DesignCAD home page on the World Wide Web. Click the WEB SITE button in the About DesignCAD dialog box. This will start your browser and call up our Web page. The browser will automatically load the site's URL or address.

Getting System Information

DesignCAD can tell you some basic information about your computer system. Click the SYSTEM INFO button in the About DesignCAD display box. A message box appears with information about your operating system, memory and free space on your hard drives.

Add Menu Item Command

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	ADD MENU ITEM
Point 1:	Point inside the menu area

The Add Menu Item command adds a command to an existing digitizer menu.

Using the Command

Open the digitizer menu to be changed. Choose the ADD MENU ITEM command from the DIGITIZER submenu of the TOOLS menu. Set a point in the area the new command is to occupy. The Digitizer Menu Command dialog box appears. Enter the name of the new command in the COMMAND box.

Click CONTINUE to add another command or DONE to close the dialog box.

See Also: *Close Digitizer Menu Command, Create Digitizer Menu Command, Load Digitizer Menu Command, Remove Menu Item Command, Save Digitizer Menu Command*

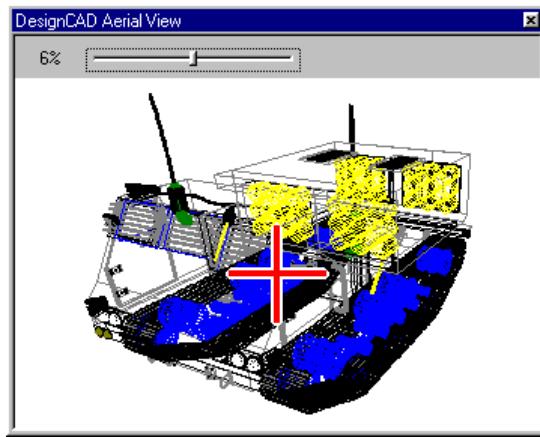
Aerial View Command

Menu:	VIEW
Menu Command:	AERIAL VIEW

The Aerial View command makes it possible to see the part of the drawing you're working on relative to the entire drawing, even when you've zoomed in closely to a specific portion of the drawing.

Using the Command

Select the AERIAL VIEW command from the VIEW menu. The Aerial View window appears.



The Aerial View command provides a bird's-eye view of the drawing, even when the view window is zoomed in on the drawing. The Aerial View window can be sized and placed anywhere on the screen.

The focus of the active view window can be changed using the Aerial View window. To change the focus of the active view window, just click on the red crosshairs in the Aerial View window with the mouse. Now click on the area in the Aerial View window that you want to be the center of the active view window.

Align Drawing Command

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	ALIGN DRAWING
Point 1:	Point on the digitizer pad
Point 2:	Corresponding point on the screen
Point 3:	Point on the digitizer pad
Point 4:	Corresponding point on the screen

The Align Drawing command synchronizes the digitizer pad with the display quickly and easily.

Using the Command

Choose ALIGN DRAWING from the DIGITIZER submenu of the TOOLS menu. Set a point of reference on the digitizer pad. Set a second point with regard to the drawing screen that will correspond to the first point set on the digitizer pad. Now set a third point as another point of reference on the digitizer pad. Finally, set the fourth point with regard to the drawing screen; this point will correspond to the third point which was set with respect to the digitizer pad. This will tell DesignCAD the relationship that the area of the digitizer pad should have with the drawing screen.

Note: DesignCAD will ignore this command if you do not have a digitizer or if it is not recognized by your computer.

See Also: *Digitizer Tracing Mode*

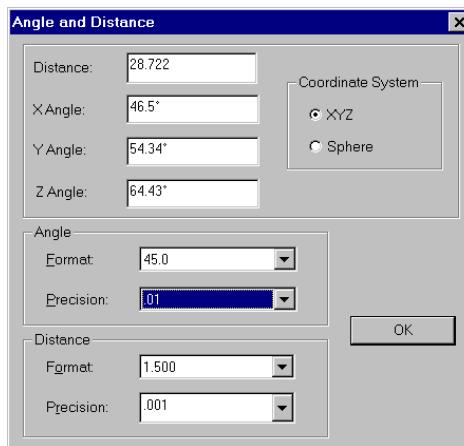
Angle & Distance Between Points Command

Menu: DIMENSION
 Submenu: INFO
 Menu Command: ANGLE & DISTANCE BETWEEN POINTS
 Point 1: First point to be calculated
 Point 2: Second point to be calculated

The Angle & Distance Between Points command calculates the angle and distance between two points and displays that information in a dialog box. You can then insert the measurements into your drawing.

Using the Command

Choose the ANGLE & DISTANCE BETWEEN POINTS command. Set two points for the angle and distance to be measured. A dialog box appears, showing the results, with the distance first and the angles second.

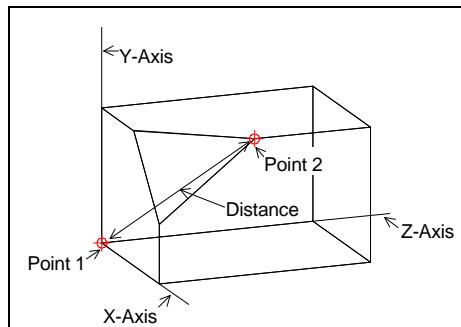


XYZ Coordinate System

When the XYZ Coordinate System is selected, this command uses the location of the first point as the origin for the angular measurements.

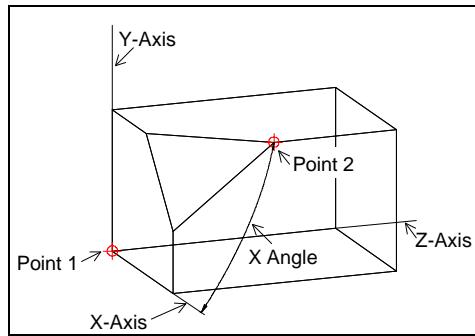
Distance

The Distance is the linear distance between the two points.

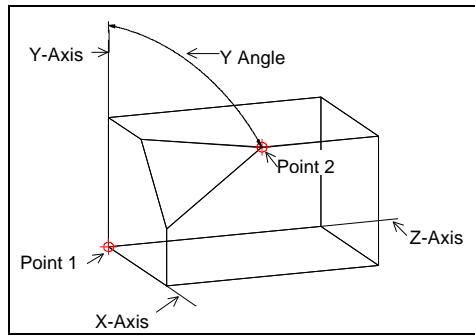


X Angle

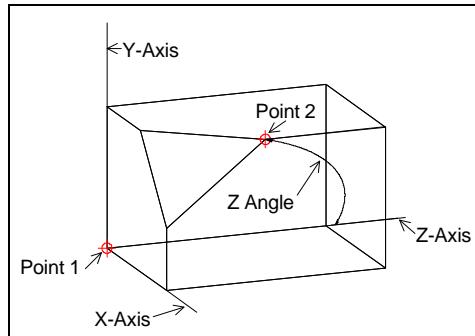
The X Angle is the angle from the X-Axis to the second point.

**Y Angle**

The Y Angle is the angle from the Y-Axis to the second point.

**Z Angle**

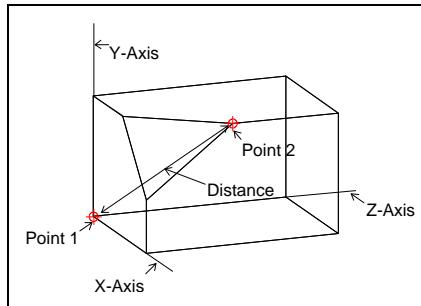
The Z Angle is the angle from the Z-Axis to the second point.



Sphere Coordinate System

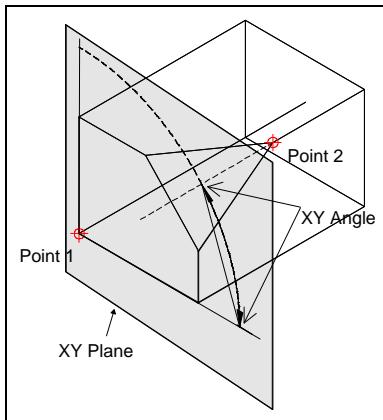
Distance

The Distance is the linear distance between the two points.



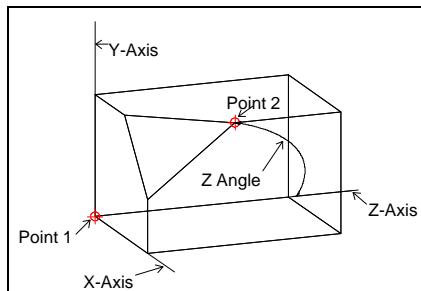
XY Angle

The XY Angle is the angle from Point 1 to Point 2 relative to the XY Plane. In other words, if the XY plane were a projector screen, and the front view of the drawing was projected on to that screen, the angle would be measured from Point 1 to Point 2 on the screen.



Z Angle

The Z Angle is the angle from the Z-Axis to the second point.



Angle

Format

This option changes how the measurement is displayed. DesignCAD lets you choose from the following format options:

- degrees
- grads
- radians
- degrees, minutes, seconds
- geographical angles

Precision

This option lets you choose the degree of accuracy that DesignCAD uses to display the angle information.

Distance

Format

Use this option to select the format you want DesignCAD to use for displaying distance information. The following options are available:

- decimal
- fractional
- engineering
- architectural

Precision

This option lets you select the degree of accuracy DesignCAD uses to display the distance information, in whole numbers or fractions.

Hint: DesignCAD lets you copy and paste the calculations into your drawing.

Example: Calculate the angle and distance of two points and insert the result into a drawing.

Choose the ANGLE & DISTANCE BETWEEN POINTS command. Set two points for the angle and distance to be measured. A box appears, giving the results of the calculation. Choose the FORMAT and PRECISION options you want.

Next, select the results of the calculation. Press **Ctrl+C** to copy the angle and distance to the Clipboard. Return to the drawing and choose the TEXT command. Move into the TEXT box in the dialog box by pressing the **Tab** key or by clicking there with the mouse. Now press **Ctrl+V** to paste the angle and distance of the points into the TEXT box.

Once you have pasted the results, you can enter the text as you normally would, by returning to the drawing and setting points where you want to place the text.

Angle Between Two Lines Command

Menu: DIMENSION

Submenu: INFO

Menu Command: ANGLE BETWEEN TWO LINES

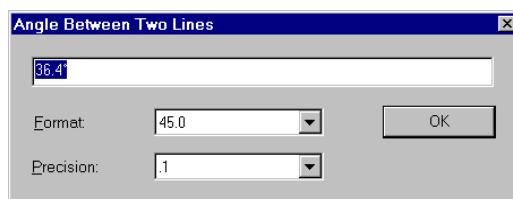
Point 1: The first line of the angle to be measured

Point 2: The second line of the angle to be measured

The Angle Between Two Lines command measures the angle between two non-parallel lines and displays the result. This measurement can then be copied and pasted into a drawing (see Angle & Distance Between Points command).

Using the Command

Choose the ANGLE BETWEEN TWO LINES command. Set a point on the first line of the angle to be measured. Set a point on the second line. The result is displayed in the Angle Between Two Lines dialog box.



Click the OK button to close the dialog box.

Angle Between Two Lines

Format

This option changes how the measurement is displayed. DesignCAD lets you choose from the following format options:

- degrees
- grads
- radians
- degrees, minutes, seconds

Precision

This option lets you choose the degree of accuracy that DesignCAD uses to display the angle information.

Example: Calculate the angle between two lines.

Choose the ANGLE BETWEEN TWO LINES COMMAND. Set a point on each of the two lines to be used for calculating the angle. After the Angle Between Two Lines box appears, click on the FORMAT down arrow and choose one of the four options. Next, select the precision you want in the PRECISION box. DesignCAD immediately shows the result in the selected format and precision. Click the OK button to close the dialog box.

Angle Constraint Mode Command

Shortcut Key: **Ctrl+F4**

Toolbox Icon: 

Angle Constraint Mode restricts point placement in drawing commands to a specified set of angles from the previous point. The set of angles used in Angle Constraint Mode may be set using the Angle Constraint Settings Command.

Using the Command

Activate ANGLE CONSTRAINT MODE. Select a drawing command and set a beginning point. Set the number of points required or desired to draw the entity. Angle Constraint Mode will remain active until the command is selected again to turn it off.

Example: Draw a line with segments at 0, 30, 90, 150, 180, 210, 270, or 330 degrees.

Activate ANGLE CONSTRAINT MODE. Select the LINE command and set a point on the screen. Move the cursor around slowly. Notice that the rubber-band line only moves in one of eight directions. Set a point for the body of the line. Slowly move the cursor around. Again, notice that the rubber-band line only moves in one of eight directions. Set another point. Press **Enter** to complete the command and add the line to the drawing.

See Also: Angle Constraint Settings Command

Angle Constraint Settings Command

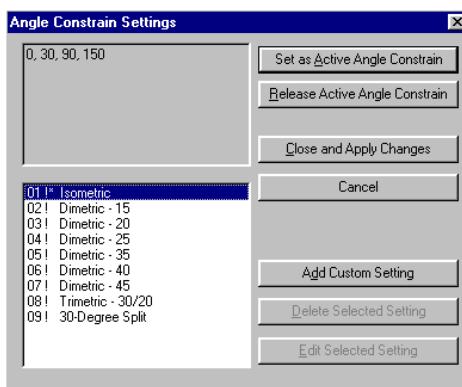
Menu: **OPTIONS**

Menu Command: **ANGLE CONSTRAINT SETTINGS**

This command is used to change the settings used in Angle Constraint Mode. Default settings may be used or custom settings may be added and used.

Using the Command

Select the ANGLE CONSTRAINT SETTINGS command. The Angle Constraint Settings dialog box appears.



Set as Active Angle Constraint

Clicking this button sets the selected set of angle constraints as the current one.

Release Active Angle Constraint

Clicking this button disables all angle constraints.

Close and Apply Changes

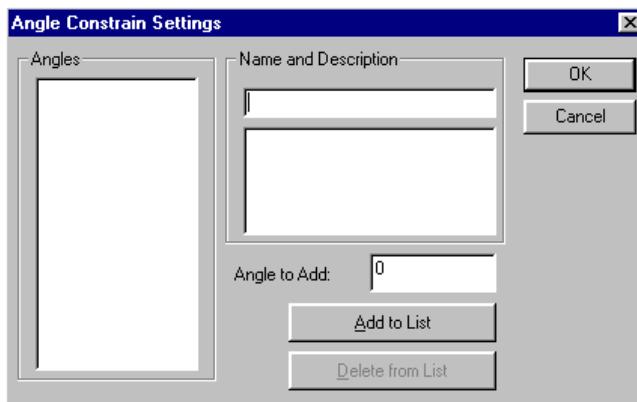
Clicking this button closes the dialog box. DesignCAD is ready to use the selected set of angle constraints as the current one.

Cancel

Clicking this button closes the dialog box. DesignCAD is ready to continue using the set of angle constraints that was being used before the dialog box was opened.

Add Custom Setting

Clicking this button opens a dialog box that lets you add sets of custom angle constraints.

**Angles**

This area contains a list of the angles that will be used in the new set of angle constraints.

Name and Description

This area contains the name and description that will be used for the new set of angle constraints.

Angle to Add

Enter an angle in this box and then click on the ADD TO LIST button to add it to the angle constraint set.

Add to List

Once an angle has been entered in the ANGLE TO ADD box, click this button to add it to the set of angle constraints.

Delete from List

Click this button to remove the selected angle measurement from the list of angles in the Angles area.

OK

Clicking this button saves the changes you have made and returns you to the Angle Constraint Settings dialog box.

Cancel

Clicking this button disregards the changes you have made and returns you to the Angle Constraint Settings dialog box.

Delete Selected Setting

Clicking this button deletes the currently selected set of angle constraints. Only custom sets of angle constraints may be deleted.

Edit Selected Setting

Clicking this button opens the currently selected set of angle constraints so that it may be edited. Only custom sets of angle constraints may be edited.

Animation Mode Command

Menu:	ANIMATION
Menu Command:	ANIMATION MODE

The Animation Mode command is used to animate or add motion to objects in a DesignCAD drawing.

This command is also used to produce animated AVI video files that can be played by Windows' Media Player and other video player applications. Animation Command will also produce VRML files, which can be used to animate a drawing for display on the Internet.

DesignCAD animation works on the same principle used by motion pictures and animated cartoons. A series of drawing images are recorded one frame at a time, with the position of the animated object changed slightly from frame to frame. When all the frames are displayed in rapid succession, the animated drawing creates the illusion of motion and continuity for the viewer.

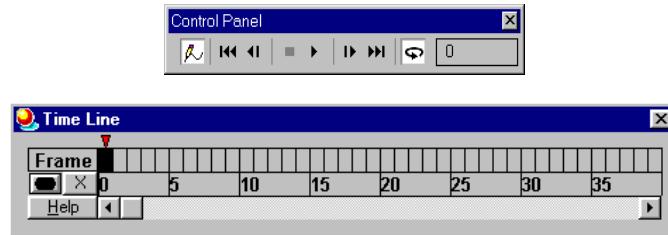
DesignCAD animation is performed by establishing keyframes of a drawing at the start, middle and end of an animated sequence. These keyframes serve as guide posts for the sequence. You must select how much time (how many frames) will pass between keyframes. You must also manipulate and move the drawing object between keyframes. This tells DesignCAD what and how much "movement" to display in relation to various times in the animation sequence.

Using the Command

You should have a drawing loaded when you choose the Animation Mode Command.

Hint: Pay attention to where the selection handles on the animated object are located. The selection handle serves as a reference point for all the movement, rotation, or scale changes that will occur in your animation.

With the drawing open, choose ANIMATION MODE from the ANIMATION menu. The Control Panel and Time Line dialog boxes appear.



To set the first keyframe, make sure the box above the zero is highlighted in the Time Line (there will be an arrowhead above it) and click the SET KEYFRAME button .

Begin to set the next keyframe by clicking in the box above the number for the next frame that will be a keyframe. Notice that the arrowhead moves above the box you clicked. The boxes that correspond to the keyframes you set (except for the 0 frame) are filled with red, indicating that they are keyframes.

The number of frames left between keyframes determines the size of the animation increments in the final animation. In other words, if you are rotating an object 90 degrees between two keyframes that are ten frames apart (frames 0 and 9), there will be 9 movements. In the final animation, the object will rotate 10 degrees per frame (90 divided by 9 is 10).

Hint: Animation runs at about 15 frames per second, so 30 frames would equal approximately two seconds worth of animation.

Now enter all pertinent motion information for your animated object. This is the amount and type of movement you want to occur from the first keyframe to the second keyframe. You can use the Move, Rotate, Scale or use any combination of these commands.

Once you have moved the object in the desired manner, verify that you have highlighted the box above the number you want to set as the second keyframe and click the SET KEYFRAME button to lock in the keyframe.

Repeat this series of actions as many times as necessary to complete your animation. If you make a mistake and want to delete a keyframe, in the Time Line highlight the box above the keyframe you want to delete and click on the DELETE KEYFRAME button .

Note: Don't forget to record a keyframe any time you change the direction of movement, the direction of a rotation, etc. For example, if you are going to move an object to the left and then up in your animation, make sure you record a keyframe before you move the object up. If you don't record a keyframe after you move the object to the left but wait until after you have

moved the object to the left and then up, DesignCAD doesn't know that you wanted to show movement to the left and then upward movement. The object will move diagonally when the animation is played.

Playing and Saving An Animation Sequence

-  To show or hide the paths of animated objects, click on the SHOW PATH button.
-  To rewind your animation, click on the REWIND button.
-  Click on the BACK ONE FRAME button to move the animation backward one frame.
-  Click the STOP button to stop an animation that is playing.
-  Click the PLAY button on the Animation toolbox to play and examine the animation.
-  Click the FORWARD ONE FRAME button to move the animation forward one frame.
-  To fast forward to the end of the animation, click on the FAST FORWARD button.
-  To play the animation over and over again, click on the LOOP button.

If you are finished, save your work as an animation template. Select the SAVE ANIMATION TEMPLATE command from the ANIMATION menu. Enter a name for the animation in the dialog box and then click OK. The animation template will be saved as a part of the drawing file.

Playing Animation in DesignCAD

To play an animation, display the drawing you animated, then choose ANIMATION|LOAD ANIMATION TEMPLATE. Select the template file that animated the drawing, and click OK. You can now click the play button to run your animation.

Hint: The Animation Toolbar remains in place until you deselect the ANIMATION MODE command under the ANIMATION menu.

See Also: **AVI Command**, **VRML Command**, **Walk Through Mode Command**

Arc Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	ARC
Shortcut Key:	A
Toolbox Icon:	

- Point 1: Center of the arc
- Point 2: Start of the arc
- Point 3: Orientation of the arc (optional)

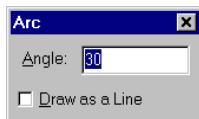
The Arc command can be used to draw an arc which spans a given angle.

Using the Command

Choose the ARC command from the Main Toolbox. Enter the arc angle in the ANGLE field in the dialog box. Finally use the mouse to specify a center point, a starting point, and an optional third point to orient the arc in 3-D space.

There are two choices for the format of the arc, which you can set in the dialog box:

1. **Arc:** Stored as an actual arc in the drawing (default setting).
2. **Draw as a Line:** Drawn as line segments that follow the shape of an arc.



The Arc format saves the arc as an Arc entity. The Line format saves the arc as a series of short line segments. You should normally use the Arc format because it is more efficient and precise. You can use the Line format if you need to distort the arc by scaling or stretching.

Example 1: Draw an arc in the XY plane with a radius of 10 and an angle of 60 degrees.

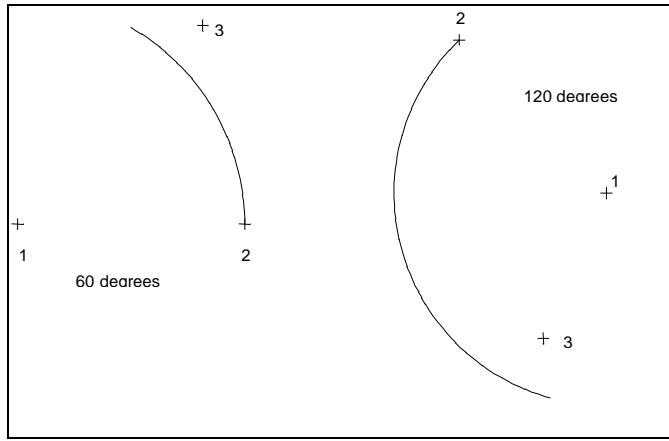
Change the view setting to the FRONT VIEW. Choose the ARC command and enter **60** degrees in the ANGLE box in the dialog box. Set a point anywhere on the screen for the arc's center. Choose the POINT RELATIVE command from the POINT menu or press the ' key (single quote), and enter **10** for DX. Set a third point anywhere on the screen with the mouse or arrow keys. Your arc should look like the one in the illustration below.

Example 2: Create a 120-degree arc similar to the one in the figure.

Set the angle as described above, and place the points as shown in the illustration below.

Example 3: Create a third arc at an angle of 60 degrees which is parallel to the XZ plane.

Set the angle as above. Set the center point, then a point for the radius. For the third point press the single quote key (' for the POINT RELATIVE command, and set DZ to **10**. In the Front view this arc appears to be a straight line. You will have to change to a different view to see it as an arc. In the Top view, this arc should look like the 60-degree arc in the illustration.



See Also: [Arc \(3-Point\) Command](#), [Arc \(Center, Begin, End\) Command](#), [Arc \(Endpoints, Center\) Command](#), [Arc \(Radius, Begin-End\) Command](#), [Tangent Arc Command](#)

Arc (3-Point) Command

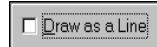
Menu:	DRAW
Submenu:	ARCS
Menu Command:	ARC (3-POINT)
Toolbox Icon:	
Point 1:	Beginning of the arc
Point 2:	A point along the arc
Point 3:	Endpoint of the arc

The Arc (3-Point) command draws an arc using a specified starting point, a second point through which the arc passes, and the endpoint of the arc.

Using the Command

You have two choices for the format of the arc in the dialog box:

- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Drawn as line segments that follow the shape of an arc.

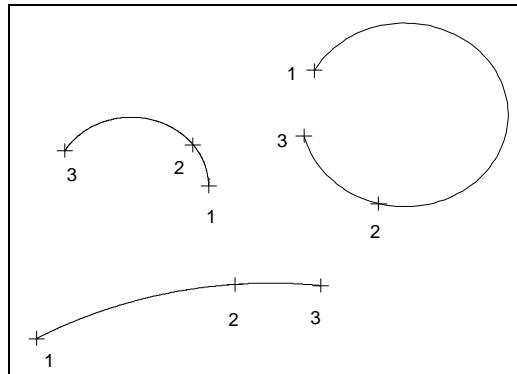


Click the Draw as a Line box if you want to change the format from Arc to Line. The program remembers your most recent selection the next time you use the Arc (3-Point) command.

The Arc format stores a center point, endpoint, and orientation point in the drawing. The Line format stores a series of points for the line approximating the arc. You can draw arcs with this command in either a decreasing or increasing angular direction. This means that the arc can be drawn in either direction from its starting point.

Example: Draw an arc passing through three points.

Select the ARC (3-POINT) command and set a point anywhere on the screen. Set a second point about an inch away (relative only to your screen) from the first in any direction. This point determines the direction that the arc is drawn. It is also a point on the arc. As you move the cursor notice that a rubber-band arc forms a representation of the arc using the cursor location as the endpoint. Set a third point for the end of the arc.



See Also: **Arc Command, Arc (Center, Begin, End) Command, Arc (Endpoints, Center) Command, Arc (Radius, Begin-End) Command, Tangent Arc Command**

Arc (Center, Begin, End) Command

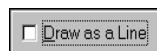
Menu:	DRAW
Submenu:	ARCS
Menu Command:	ARC (CENTER, BEGIN, END)
Toolbox Icon:	
Point 1:	Center of arc
Point 2:	Beginning of arc
Point 3:	End of arc

The Arc (Center, Begin, End) command can be used to draw an arc using the center, beginning, and end points of the arc.

Using the Command

You have two choices for the format of the arc:

- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Drawn as line segments that follow the shape of an arc.



Click the Draw as a Line box to change the format from Arc to Vector. The program retains the most recent selection each time you use the Arc (Center, Begin, End) command in the current drawing session.

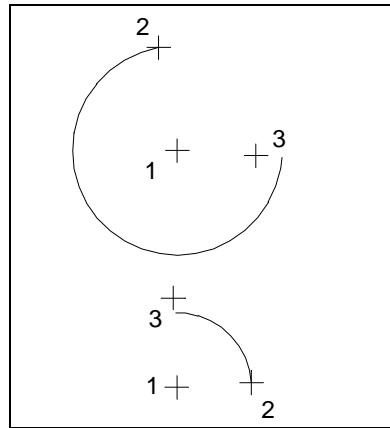
The Arc format stores a center point, endpoint, and orientation point in the drawing. The Line format stores a series of points for the line approximating the arc.

The first point you set becomes the center of the arc. The second point sets both the radius and the start angle of the arc. The third point sets the ending angle of the arc, and will not necessarily lie on the endpoint of the actual arc that is drawn.

Note: Arcs drawn with the Arc (Center, Begin, End) command are created in such a way that the arc can only span a positive angle from its starting point, so the arc will be drawn counterclockwise from point 2 to the span angle on the arc determined by point 3.

Example: Draw an arc using a specific center point.

Select the ARC (CENTER, BEGIN, END) command and set a point for the center of the arc. Set a second point about an inch directly above the first point. As you move the cursor around the screen, a rubber-band arc forms a representation of the arc using the cursor position as the span angle. When the arc spans the angle you want, set the third point.



See Also: [Arc Command](#), [Arc \(3-Point\) Command](#), [Arc \(Endpoints, Center\) Command](#), [Arc \(Radius, Begin-End\) Command](#), [Tangent Arc Command](#)

Arc (Endpoints, Center) Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	ARC (ENDPOINTS, CENTER)
Toolbox Icon:	
Point 1:	Beginning of the arc
Point 2:	End of the arc
Point 3:	Center (radius) of the arc

The Arc (Endpoints, Center) command draws an arc using points set for the beginning, end, and radius of the arc. This command is similar to the Arc (Radius, Begin-End) command except the radius is determined by the third point rather than by entering a radius in the dialog box.

Using the Command

You have two choices for the format of the arc:

- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Drawn as line segments that follow the shape of an arc.



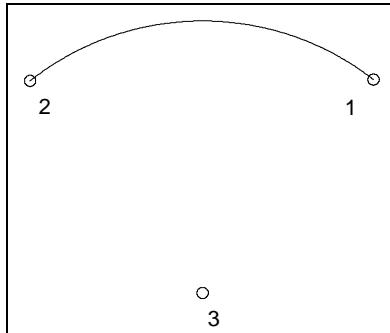
Click the Draw as a Line box to change the format from arc to vector. The program retains the most recent selection each time you use the Arc (Endpoints, Center) command in the current drawing session.

The Arc format stores a center point, endpoint, and orientation point in the drawing. The Line format stores a series of points for the line approximating the arc.

The first point determines the beginning of the arc. The second point defines the end of the arc. After the second point is set, a rubber-band arc shows how the arc will be drawn. Set a third point for the center of the arc. An arc is then drawn counterclockwise from the first point to the second point.

Example: Draw an arc with a point set for the radius.

Select the ARC (ENDPOINTS, CENTER) command. Set a point for the beginning of the arc on the right side of the screen. Set a second point for the end of the arc on the left side of the screen. After this point is set, a rubber-band arc will be drawn as the cursor is moved. Set a third point for the radius in the center of the screen, below the first two points. An arc will be drawn from the first point to the second point, using the third point to determine the radius.



See Also: [Arc Command](#), [Arc \(3-Point\) Command](#), [Arc \(Center, Begin, End\) Command](#), [Arc \(Radius, Begin-End\) Command](#), [Tangent Arc Command](#)

Arc (Radius, Begin-End) Command

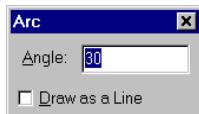
Menu:	DRAW
Submenu:	ARCS
Menu Command:	ARC (RADIUS, BEGIN-END)
Toolbox Icon:	
Point 1:	Start of arc
Point 2:	End of arc
Point 3:	Orientation

The Arc (Radius, Begin-End) command can be used to draw an arc of pre-determined radius by specifying the beginning and ending points of the arc and setting an optional third point to orient the arc in 3-D space.

Using the Command

Like the Arc command, the Arc (Radius, Begin-End) command offers two choices for the format of the arc:

- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Drawn as line segments that follow the shape of an arc.



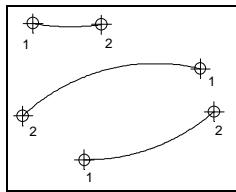
The Draw as a Line box in the dialog box is used to change the format from Arc to Vector. The desired radius is entered in the RADIUS box. The settings you choose will be used as the default settings for that command in the current drawing until you reset them.

The Arc format stores a center point, endpoint, and orientation point in the drawing. The Line format stores a series of points for the line approximating the arc.

The Arc (Radius, Begin-End) command automatically creates an arc with a span of less than 180 degrees, the smallest possible arc based on the supplied radius and the first two points set. To create an arc with a span greater than 180 degrees, set the third point inside the area enclosed by the rubber-band arc and the first two points. The arc is drawn in an increasing angular direction from the first point to the second.

Example: Create an arc with a radius of 20.

Select the ARC (Radius, Begin-End) command. Enter the RADIUS as **20** in the dialog box. Set a point somewhere around the center of the screen. Move the cursor up and to the right a couple of inches and set a second point. Now, move the cursor between the two points. Notice that DesignCAD forms a rubber-band arc. Set a third point to determine the orientation of the arc. The arc is inserted into the drawing.



See Also: **Arc Command, Arc (3-Point) Command, Arc (Center, Begin, End) Command, Arc (Endpoints, Center) Command, Tangent Arc Command**

Area Command

Menu: DIMENSION

Submenu: INFO

Menu Command: AREA

Points: Determine the area to be calculated

The Area command calculates the area of part of a drawing and displays the results in a dialog box. This information can be inserted into the drawing.

Using the Command

To use the command, select the AREA command from the INFO submenu of the DIMENSION menu. Then set points around the area you want to measure. The points will be connected with a temporary line. The area of the shape will be calculated and displayed on the screen.



The area is calculated using the current Drawing Units. If the first and last points are not in the same location, an imaginary line connects those two points to close the shape.

Show Results in Acres

Select the SHOW RESULTS IN ACRES box if you want DesignCAD to calculate the area in acres instead of Drawing Units.

Precision

Click the down arrow in the PRECISION box to change how precisely DesignCAD displays the information. You can set the accuracy to round the result anywhere from the nearest 10 millions to the nearest 10 millionths.

Inserting Area into a Drawing

In the AREA box, highlight the area text. Press **Ctrl+C** to copy the text to the Clipboard.

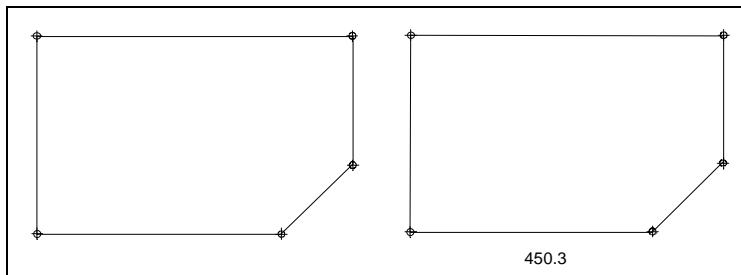
68.047

Next, click the **OK** button. Choose the **TEXT** command from the Main Toolbox. Click in or **Tab** into the **TEXT** box in the dialog box. Press **Ctrl+V** to paste the Clipboard contents. Set the points for the text.

Example: Calculate the area of a closed entity and insert the figure into the drawing.

Draw a closed entity. Select the **AREA** command from the **INFO** submenu of the **DIMENSION** menu. Set points to designate the area to be calculated. After the first point is set, a rubber-band line shows the area to be calculated. Press **Enter**. An information box shows the area in Drawing Units.

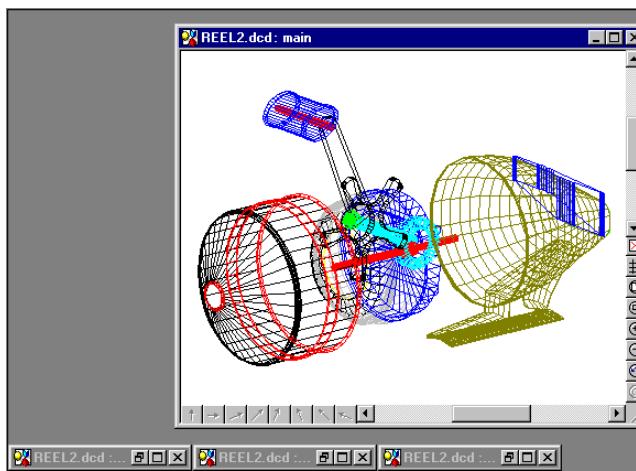
In the **AREA** dialog box, highlight the area text. Press **Ctrl+C** to copy the text to the Clipboard. Click the **OK** button and choose the **TEXT** command. Click in or **Tab** into the **TEXT** box in the dialog box. Press **Ctrl+V** to paste the Clipboard contents. Place the text as you normally would, by setting points in the drawing for the text.



Arrange Icons Command

Menu:	WINDOW
Menu Command:	ARRANGE ICONS

The **Arrange Icons** command organizes the view icons at the bottom of the window.



Using the Command

Any viewing window in DesignCAD 3D MAX can be minimized, maximized, or sized. If you have minimized several view windows into icons, you can arrange them at the bottom of the drawing screen. Choose the ARRANGE ICONS command from the WINDOW menu. The icons are automatically arranged.

Array Command

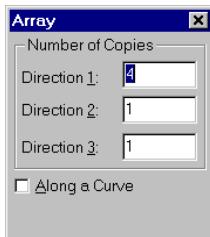
Menu:	DRAW
Menu Command:	ARRAY
Toolbox Icon:	
Point 1:	Location and direction of first copy (relative to handle)
Point 2:	Axis of Direction 2 (relative to handle on the original)
Point 3:	Axis of Direction 3 (relative to handle on the original)

The Array command copies a selected object a specified number of times in as many as three directions. The object you want to copy must be selected before you use this command.

Using the Command

The Array command can make multiple copies of an object in one dimension (in a single row), in two dimensions (rows and columns), or in three dimensions (rows, columns, and layers).

To copy in a single direction or dimension, enter the number of copies in DIRECTION 1, and enter 1 for DIRECTION 2 and DIRECTION 3. Set a point for the location of the second copy of the object. (The first copy is the original.) The copy is positioned so that the point that corresponds to the selection handle on the original is located on that point. Subsequent copies are placed at the same direction and distance as the first point from the selection handle of the original. In other words, all copies are evenly spaced based on the spacing of the first two copies.



The Array command can also make copies in two dimensions—rows and columns. To do this, enter the number of columns in DIRECTION 1, and the number of rows in DIRECTION 2. Leave DIRECTION 3 set to 1. Set a point for the offset (the distance and direction) of copies in the first row. Then set a second point for the offset of the rows themselves. For example, you might want to set the first point to the right of the selection handle, and the second point directly above the selection handle.

To make a three-dimensional array with this command, add the number of "layers" of rows and columns in DIRECTION 3, and add a third point for the offset of each layer.

Along Curve

The Array command can also draw an array along a curve. To draw an array along a curve, check the ALONG A CURVE box in the dialog box. An additional option is added to the dialog box: Perpendicular. Check the Perpendicular box to draw the objects in the array perpendicular to the curve.

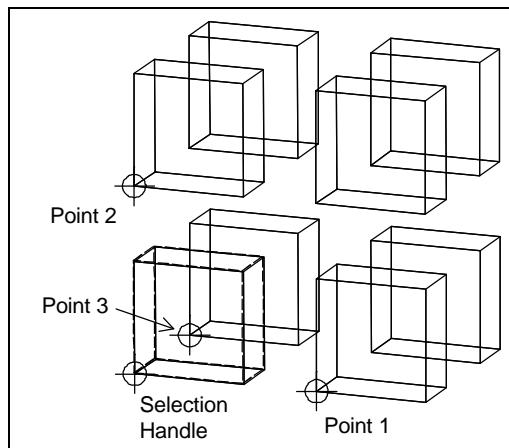
To specify exactly what is set perpendicular to the curve, it is necessary to set three selection handles for the object to be arrayed. If a plane were drawn using the three selection handles as points, the plane is what would be set perpendicular to the curve.

Set the number of copies to be drawn along the curve in the DIRECTION 1: box in the dialog box. Set a point on the curve the array is to be drawn along.

Hint: The easiest way to set the point on the curve for the array is to use the Gravity command to snap to one of the endpoints of the curve.

Example: Make two rows of two copies of a box, in three dimensions.

Select the box and choose the ARRAY command. Enter **2** for each direction (DIRECTION 1, DIRECTION 2, and DIRECTION 3). Set a point directly to the right of the selection handle, and a second point above it. Finally, set a point on the third axis, away from the handle on the original box. The box will be duplicated in a $2 \times 2 \times 2$ formation.



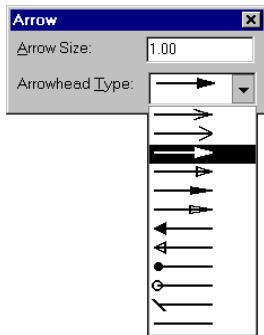
Arrow Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	ARROW
Shortcut Key:	>
Toolbox Icon:	
Point 1-n:	Body of the arrow
Point n:	Arrowhead

The Arrow command draws an arrow using two or more points.

Using the Command

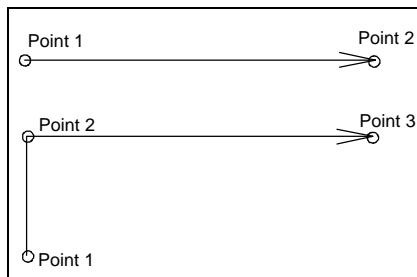
Set the first point at the tail and work toward the arrowhead. Use the options in the dialog box to set the arrowhead size and type.



Note: The Arrowhead Size option is actually a ratio of the current text size, thus the same Arrowhead Size will only make the arrowhead for a second arrow the same size if the text size has not changed since the first arrow was drawn.

Example: Draw an arrow.

Select the ARROW command and set 1 for the ARROW SIZE option in the dialog box. Select the arrow type. Set a point for the tail of the arrow and two points for the body of the arrow, so that they form a right angle. Press **Enter** to draw the arrow.



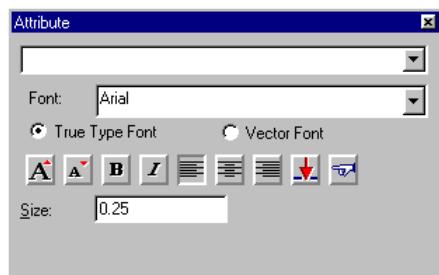
Attribute Command

Menu:	DRAW
Menu Command:	ATTRIBUTE
Shortcut Key:	\$
Toolbox Icon:	
Point 1:	Location of the attribute

The Attribute command is used to assign information to objects in the drawing. Later you can use the Material List command to list the types and numbers of attributes you have placed in your drawing.

Using the Command

Select the ATTRIBUTE command. Click the TEXT box in the dialog box and enter the text. The Text box keeps a history of the last 20 entries. To use this feature, click the down arrow on the right end of the box and highlight the entry to be used again.



Select the font for the text from the FONT list box. If the TRUE TYPE FONT option is chosen, only True Type fonts are available in the Font list box. If the VECTOR FONT option is chosen, only Vector fonts are available in the Font list box.

Click the INCREASE TEXT SIZE  button to increase the text size.

Click the DECREASE TEXT SIZE  button to increase the text size.

Click the BOLD  button to make the text bold.

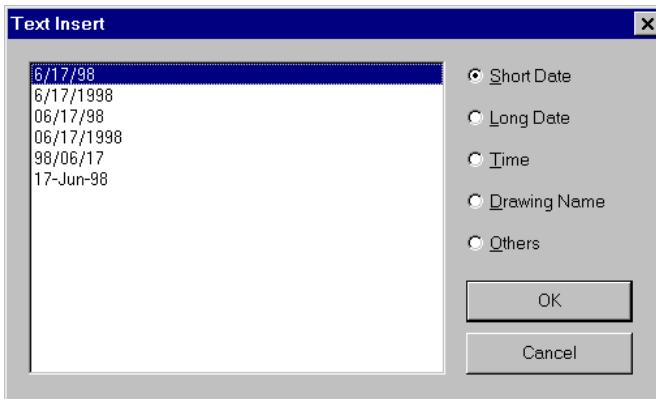
Click the ITALIC  button to italicize the text.

Click the LEFT JUSTIFY  button to align the text on the left.

Click the CENTER JUSTIFY  button to center the text.

Click the RIGHT JUSTIFY  button to align the text on the right.

Click the INSERT button  to display the Text Insert box.



Choose the kind of text you would like to have automatically inserted from the list of items to the right of the Text Insert dialog main window: Short Date for example. Using the mouse, click the format for the item from those displayed in the main window of the Text Insert dialog and then click the OK button.

 Click the SAME AS button and then click on a line of text in the drawing to apply the same options to the new text.

You can enter the size for the text directly in the size box instead of using the Increase Text Size and Decrease Text Size buttons.

Set a point for the lower-left corner of the text. The Attribute is inserted in the drawing.

Attributes can be made visible or invisible by checking or unchecking the SHOW ATTRIBUTES option in the TEXT OPTIONS folder, available through the OPTIONS command in the OPTIONS menu. Leaving Attributes hidden makes working on your drawing easier.

Example: Label an object in your drawing.

Make sure that the Show Attributes option in the Text Option folder has a check next to it, showing that it is enabled. Select the ATTRIBUTE command. Click in the TEXT box in the dialog box and enter a name for the object. Leave all of the other settings in the dialog box as they are. As you move the cursor, notice the green box following the cursor. This represents the area the text will occupy. Set a point near the object. The attribute is inserted at that point.

Automatic Rendering Command

Menu:	FILE
Submenu:	PAPER SPACE VIEW FRAME SETUP
Menu Command:	AUTOMATIC RENDERING

This command is a toggle that determines whether or not View Frames that have been set up to appear with hidden line removal or shading in Paper Space Mode will be rendered when the Paper Space Template is opened.

If Automatic Rendering is disabled, the drawing will initially display faster in Paper Space Mode because the view frames will open in wireframe, but the view frames that have hidden line removal or shading turned on, will still be previewed and printed that way.

See Also: **Paper Space Mode Command**

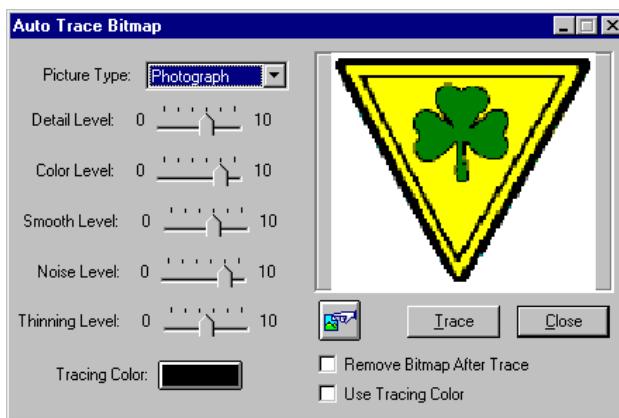
Auto Trace Bitmap Command

Menu: **TOOLS**
 Menu Command: **AUTO TRACE BITMAP**

The Auto Trace Bitmap command creates vector outlines of specified colors in bitmaps. This allows the image to be manipulated in DesignCAD. With a color or black and white scanner, a drawing or other art can be scanned and saved as a bitmap, then each color can be traced to produce a DesignCAD drawing.

Using the Command

Bring a bitmap into DesignCAD with the LOAD IMAGE FILE command. When the bitmap is displayed, select it and choose the AUTO TRACE BITMAP command from the TOOLS menu. The Auto Trace Bitmap dialog box appears.



The color to be traced is displayed on the TRACING COLOR: button. To change color, click this button. The cursor turns into an eyedropper.

Either in the thumbnail image at upper-right of the dialog box or in the main drawing area, position the eyedropper on the item you want to trace. Click the mouse button. The sample is taken from the color at the eyedropper tip. The Tracing Color: button will change to the color of the item you selected.



If there are multiple bitmaps in the drawing, you can click the SELECT BITMAP TO TRACE button, then click on the bitmap you wish to trace.

The results of an Auto Trace are affected by the Picture Type, Detail Level, Color Level, Smooth Level, Noise Level, and Thinning Level options. Experiment with these options for optimal results.

To change these values, click and hold the indicator using the mouse, then move the indicator closer to LOW or HIGH. The Auto Trace results will automatically update when one of these options is changed.

Once you have selected the bitmap to be traced and set all of the options to the desired levels, click the TRACE button. DesignCAD will trace the bitmap. If you like the results, click the CLOSE button to close the Auto Trace Bitmap dialog box.

Once you are satisfied with the result, select the bitmap and delete it. The new vector image of the bitmap remains in the drawing.

The REMOVE BITMAP AFTER TRACE option can be checked so DesignCAD will automatically delete the bitmap from the drawing once it has been traced.

Note: Auto Trace Bitmap can only trace and convert one color at a time. If the bitmap image has entities in more than one color, you must execute the Auto Trace Bitmap command for each colored item you want traced.

See Also: [Load Image File Command](#), [Scan Image Command](#)

AVI Command

Menu:	ANIMATION
Submenu:	EXPORT
Menu Command:	AVI

The AVI command exports an animation or walk through produced in DesignCAD as an AVI file.

Using the Command

Record or open an animation or walk through template. Create an AVI file from your template by clicking ANIMATION | EXPORT | AVI. Give the file a name, then click SAVE.

AVI files are very large, so DesignCAD gives you the option to save the file uncompressed or with one of six other compression ratios. Click the menu arrow and scroll to select a compression format. Select Full Frames (Uncompressed) for the best results then click OK.

See Also: [Animation Mode Command](#), [VRML Command](#), [Walk Through Mode Command](#)

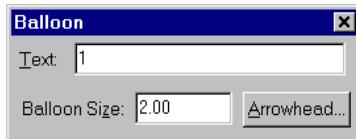
Balloon Command

Menu:	DIMENSION
Submenu:	INFO
Menu Command:	BALLOON
Toolbox Icon:	
Point 1:	Point of arrowhead
Points 2-n:	Points along arrow. Last point is center of balloon.

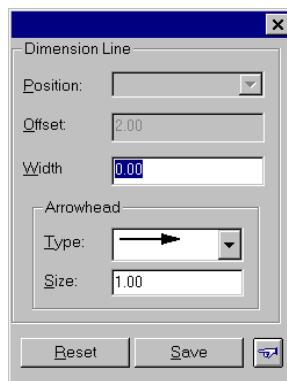
The Balloon command draws a text balloon pointing to a specific object. It is useful for identifying part numbers or item numbers in diagrams. You may specify the size of the balloon and the text to include; the text is sized to fit inside the balloon.

Using the Command

Choose the BALLOON command from the INFO submenu in the DIMENSION menu. Enter the desired text and balloon size in the dialog box.



Click on the ARROWHEAD button for a choice of arrow styles or to change the size of the arrowhead.



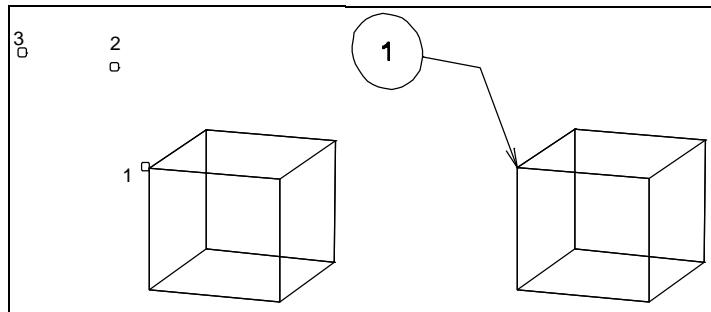
In the TYPE and SIZE boxes specify the arrowhead options you want to use. To save the options you chose to the next drawing session, click the SAVE button. Clicking the RESET button will return the options to their usual settings.

The SAME AS control  lets you match the parameters of an existing balloon. Just click on the SAME AS button, then click on the balloon already in the drawing that has the properties you want for the new balloon.

Set a point for the tip of the arrowhead, and one or more points for the body of the arrow. The last point set is used as the center of the balloon.

Example: Draw a balloon marking a box as item 1.

Choose the BALLOON command from the INFO submenu of the DIMENSION menu. Enter the TEXT: and BALLOON SIZE. Set the first point at a corner of the box. Set point 2 for the bend in the arrow, and point 3 for the text location. Press **Enter** to end the command.



See Also: Arrow Command, Pullout Command

Batch Convert Command

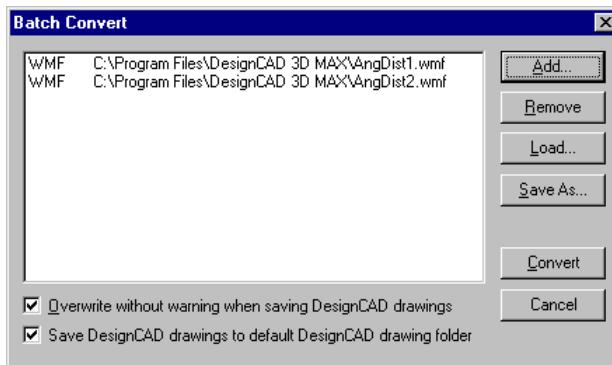
Menu: FILE

Menu Command: BATCH CONVERT

The Batch Convert command makes it possible to convert the file type of a batch or several specified files to DesignCAD format at one time.

Using the Command

Choose the BATCH CONVERT command from the FILE menu. The Batch Convert dialog box appears on the screen.



Add

This option is used to select the files to be added to a batch.

Remove

This option can be used to remove files from a batch.

Load

This option is used to load a batch (.dbx) file that has been created and saved previously.

Save As...

This option is used to save a batch (.dbx) file under a specified filename.

Overwrite without warning when saving DesignCAD drawings

This option will overwrite an existing DesignCAD drawing with the same name as a drawing that is being converted without prompting the user.

Save DesignCAD drawings to default DesignCAD drawing folder

This option places the converted DesignCAD drawings in the default DesignCAD drawing folder (this is the DesignCAD installation location unless you've changed it in the File Locations folder in the Options dialog box) when the Convert button is clicked. If this option is not checked the drawings are placed in the same directory as the files that are being converted.

Convert

Closes the Batch Convert dialog and begins converting the batch.

Close

Closes the Batch Convert dialog and returns to DesignCAD without converting files or retaining any changes.

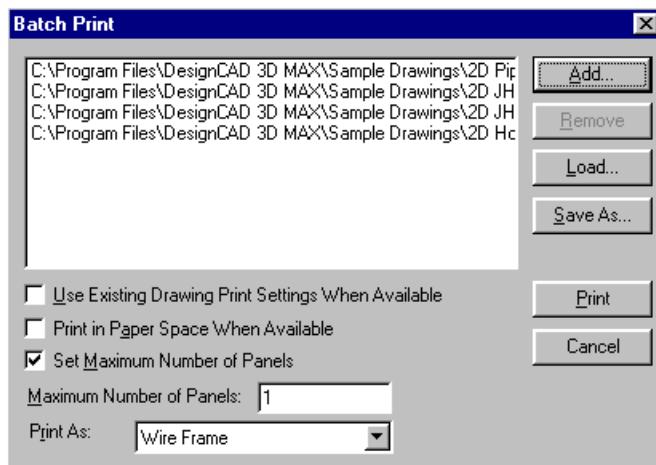
Batch Print Command

Menu:	FILE
Menu Command:	BATCH PRINT

The Batch Print command makes it possible to print a batch or several specified files at one time.

Using the Command

Choose the Batch Print command from the File menu. The Batch Print dialog box appears on the screen.

**Add**

This option is used to select the files to be added to a batch.

Remove

This option can be used to remove files from a batch.

Load

This option is used to load a batch (.dbp) file that has been created and saved previously.

Save As...

This option is used to save a batch (.dbp) file under a specified filename.

Use Existing Drawing Print Settings When Available

This option will use existing print settings, if the current print settings have been saved with the drawing by enabling the Save Current Print Settings option in the Print dialog box.

Print in Paper Space When Available

This option will print the drawing in Paper Space Mode, if a Paper Space has been configured for the drawing.

Set Maximum Number of Panels

This option makes it possible to constrain all of the files for the batch print to a specific number of panels.

Print As

This option is used to specify if the batch will be printed as Wireframe, Shading, Hide, or Hide (Pen Plotter).

Print

Closes the Batch Print dialog and sends the batch to the specified printer.

Close

Closes the Batch Print dialog and returns to DesignCAD without printing or retaining any changes.

Bearing Command

Menu: DIMENSION

Submenu: INFO

Menu Command: BEARING



Toolbox Icon:

Point 1: First point of the distance to be measured

Point 2: Second point of the distance to be measured

Point 3: Insertion point for the text

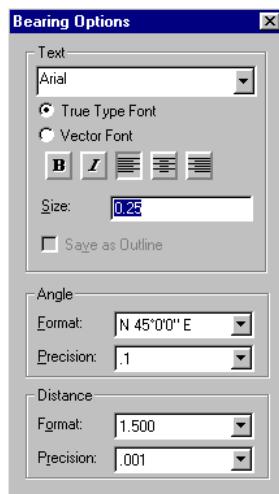
The Bearing command measures an object's bearings and inserts them into a drawing.

Using the Command

Choose the BEARING command from the Main Toolbox. Set a point at one end of the distance to be measured. Set a second point at the opposite end of the distance. The bearing text is placed in the text box of the dialog box and a rubber-band box appears at the text insertion point.



Click the OPTIONS button in the dialog box to change options for the bearing text.

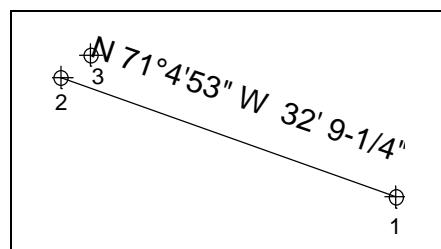


Set the desired options and press **Enter** to close the box. Position the rubber-band box where you want the text and set a final point.

Note: If you are measuring from the right side of the drawing window to the left the text will be inserted upside-down, so click the Reverse Text checkbox in the dialog box to reorient the text (insert it right-side-up).

Example: Measure and insert the bearings of a line.

Choose the BEARING command. Move the cursor close to the right endpoint of the line and right-click the mouse to set a Gravity Point. Move the cursor to the left endpoint of the line and right-click the mouse to set a second Gravity Point. Click the OPTIONS button in the dialog box to display the BEARING OPTIONS box. Set the options desired and press **Enter** to close the Bearing Options box. Click the REVERSE TEXT checkbox in the dialog box. Position the rubber-band box and set a final point to insert the bearing text.



Bezier Curve Command

Menu: DRAW
 Submenu: LINES
 Menu Command: BEZIER CURVE



Toolbox Icon:

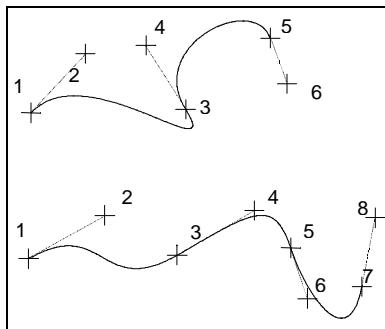
Point 1: Beginning of the Bezier curve
 Point 2: First control point for the curve (beginning tangent)
 Point 3: Pass-through point
 Point 4: Second control point (tangent for point 3)
 Point 5: Pass through point (optional)
 Point 6: Third control point (tangent for point 5)
 Point 7-n: (Optional points in pairs)

The Bezier Curve command draws a Bezier curve.

Using the Command

The first point is the endpoint of the curve, and the second point is the control point. The curve at the first point will be tangent to the second point. The third point is the endpoint, and the fourth point is the tangent. Each subsequent pair of points is a point for the curve to pass through and a control point to define the tangent at that location.

Moving a control point further from the curve makes the curve sharper. Points must be set in pairs. If you set an odd number of points, the last point is ignored.



Box Command

Menu: SOLIDS

Menu Command: BOX

Shortcut Key: 1



Toolbox Icon:

Point 1: First corner of the box
 Point 2: Opposite corner of the box

This command draws a 2-D or a 3-D box. A 2-D box is defined as a Plane. A 3-D box is defined as a Solid.

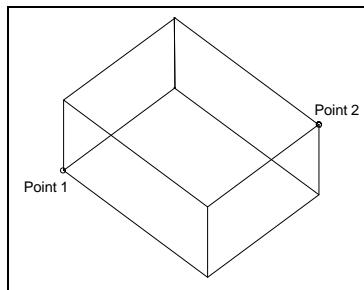
Using the Command

For a 2-D box, set a point for a corner of the box. A rubber-band box shows how the box will be drawn. Set a second point for the opposite corner.

To draw a 3-D box, set a point for one corner of the box. Move the cursor until the opposite corner of the box is where you want to place it. Then hold down the **Ctrl+Shift** keys and move the cursor in or out on the third axis. Set the second point when the box is in position.

Example: Add a 3-D box to your drawing.

Select the **BOX** command and set a point for one corner of the box. Move the cursor up and to the right of the first point, noticing the rubber-band square being drawn. When the square is the size you want, move the cursor out along the Z-axis by holding down **Ctrl+Shift** and moving the mouse up. The square turns into a 3-D box. When it's as deep as you want, set the second point to draw the box.



Break Line Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	BREAK LINE
Shortcut Key:	

The Break Line command breaks a line entity consisting of two or more line segments into separate line entities.

Using the Command

Select the entity (or entities) to be broken. Choose the **BREAK LINE** command with the shortcut key (|), or from the **EDIT | SELECTION EDIT** menu. You can use the command on a line created with the Line or Polygon commands, or created with Arcs, Circles, or Curves drawn as lines. The line selected will be broken into several separate lines, depending on how many vertices the line has. The line looks the same but is actually made of separate line entities.

Example: Break apart an object made up of two or more line segments.

Select the object. Press the | key to select the **BREAK LINE** command. Although the object looks the same, it's now broken into separate entities.

Calculator Command

Menu: TOOLS
Menu Command: CALCULATOR

The Calculator command allows you to perform calculations on the screen. Once the computation is made, you can copy the result and paste it into a drawing.

Using the Command

Choose the CALCULATOR command. The DesignCAD Calculator window appears. In the calculation box enter the values and operators for the calculation. Then press Enter or click on the COMPUTE button. DesignCAD makes the calculation and displays it in the box. Click the CLOSE button to end the command.

The expression entered in the calculation box can contain a formula or expression. The following are examples of valid expressions:

45*23

(6+23)*4

SQRT(9)

SIN(45)

The expression can contain mathematical functions and the following operators:

- + Addition
- Subtraction
- * Multiplication
- / Division
- ^ Raises a number to a power ($2 ^ 3 = 8$)

Inserting a Calculation into a Drawing

Select the text in the calculation box if it is not already selected. Press **Ctrl+C** to copy the text to the Clipboard. Click the CLOSE button to return to your drawing.

Next, choose the PASTE command. As soon as you do, a rubber-band text box appears, showing how the text will be inserted. Set a point for the lower-left corner of the text. The text is inserted into the drawing.

You can also insert the results of a calculation into a drawing by using one of the text commands which allow you to change the font, style, alignment, size, and angle. First, copy the text from the calculation box to the clipboard. Next, choose the TEXT command. Move the cursor to the TEXT box in the Command Line. Press **Ctrl+V** to paste the contents of the clipboard into the box. Set the other options as desired. Finally, return to the drawing and set the points for the text. The result of the calculation is inserted into the drawing.

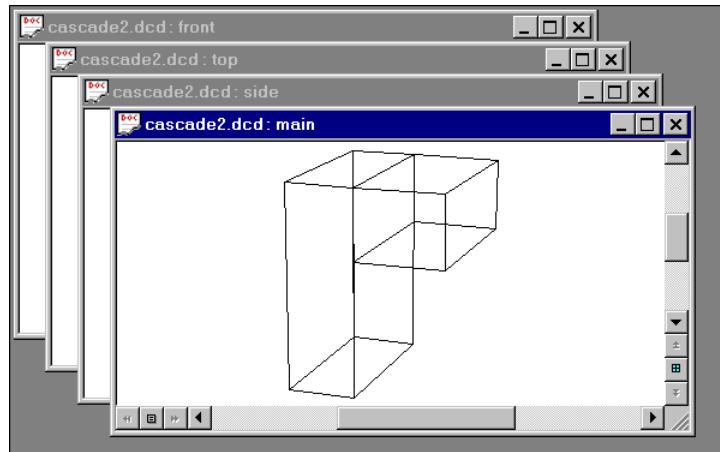
Example: Find the log of 18.

Choose the CALCULATOR command. Enter **log(18)** in the calculation box in the DesignCAD Calculator window. Press the **Enter** key or the COMPUTE button. The result, 1.26, is displayed in the box.

Cascade Command

Menu:	WINDOW
Menu Command:	CASCADE

The Cascade command arranges open drawing windows so that they overlap on the screen.



Using the Command

Choose the CASCADE command in the WINDOW menu. All open drawing windows are arranged on the screen.

See Also: *Tile Vertical Command, Tile Horizontal Command, DesignCAD Tile Command*

Center of Gravity Command

Menu:	POINT
Menu Command:	CENTER OF GRAVITY
Toolbox Icon:	
Point 1:	Point on the object

This command finds an object's center of gravity. For example, you can use the Center of Gravity command to find the volumetric center of complex solids. This command is also useful for finding the geometric center of a plane.

Using the Command

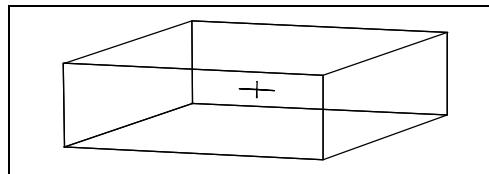
To use the Center of Gravity command, select it from the POINT menu. Then set a point on or near the object. The cursor snaps to the object's center of gravity.

Note: If you select the Center of Gravity command from the Point menu, it will always set a point.

If the Move Cursor Only  button is "pushed in" on the Snap Toolbox and you select the command from the Snap Toolbox instead of from the menu the command will just move to the position instead of setting a point there.

Example: Find the center of gravity of a solid.

Select the CENTER OF GRAVITY command from the menu. Click somewhere on the solid. The cursor sets a point at the exact center of the solid.



Chamfer Command

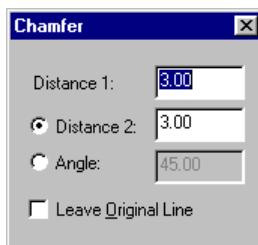
Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: CHAMFER
 Shortcut Key: **Ctrl+F**
 Toolbox Icon: 

Point 1: One of the lines on the corner to be chamfered
 Point 2: The other side of the corner

The Chamfer command cuts a segment off a corner and replaces the corner with a flat face of specified depth. This command works on line or plane entities in two dimensions. To cut off the corner of a solid object, use the Slice command.

Using the Command

Choose the CHAMFER command. Enter the chamfer depth for the first side of the chamfer in the DISTANCE 1 box. Enter the chamfer depth for the second side of the chamfer in the DISTANCE 2 box or click the ANGLE option and then enter the ANGLE for the chamfer. The values you enter will be the default values for your next chamfer.

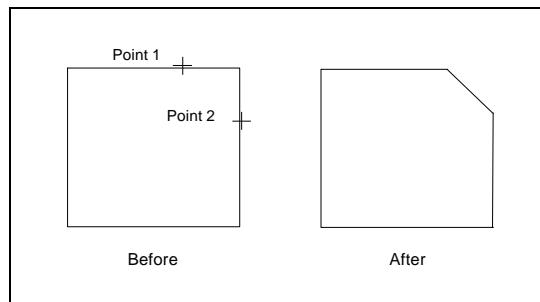


To draw the chamfer but leave the lines that formed the corner also, check the LEAVE ORIGINAL LINE box.

Next, set a point on one of the lines that forms the corner, and set another point on the other line. The corner is cut off to the specified depth.

Example: Cut one corner off a plane.

Choose the CHAMFER command and set the CHAMFER DEPTH at 3 in the dialog box. Set a point on one of the sides of the plane. Next, set a second point on an adjoining side. The corner of the plane is cut off to the specified depth.



Circle (3-Point) Command

Menu:	DRAW
Submenu:	CIRCLES
Menu Command:	CIRCLE (3-POINT)
Toolbox Icon:	
Point 1:	Any point on the circle
Point 2:	A second point on the circle
Point 3:	A third point on the circle

The Circle (3-Point) command draws a circle that passes through three points. The three points must not lie in a straight line.

Using the Command

The circle is saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the CIRCLE option is selected, the circle is saved in the drawing as a Circle entity. This is the way circles are normally saved with DesignCAD 3D MAX.

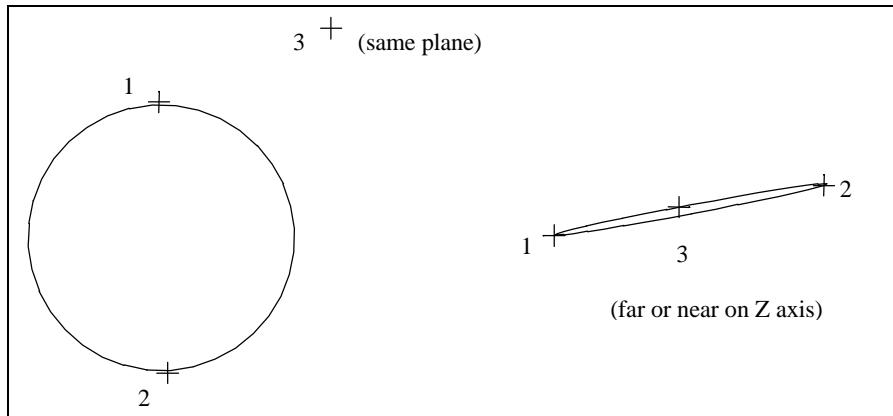
The circle can be saved as a plane so it can be shaded, subtracted, etc.

The circle can also be saved as a line. This makes it possible to scale the circle or treat it as a line entity with other commands.

Usually, you will want to save the circle as a Circle entity or as a Plane entity.

Example: Draw a circle passing through three points.

Select the CIRCLE (3-POINT) command and set a point on the screen through which the circle will pass. Move the cursor away from the first point in any direction and set a second point. Now, as you move the cursor, you see a rubber-band circle, showing you how the circle would look if you set the final point at the cursor location. Move the cursor along the Z axis by holding down **Ctrl+Shift** and moving the mouse forward or backward. This will move the cursor out or in along the Z axis respectively. The circle will pivot and change size according to the location of the third point on the Z axis. When the circle lies in the desired plane and is the correct size, set the third point.



Circle (Center, Outside) Command

Menu:	DRAW
Submenu:	CIRCLES
Menu Command:	CIRCLE (CENTER, OUTSIDE)
Shortcut Key:	O (the letter, not the number)
Toolbox Icon:	
Point 1:	Center of the circle
Point 2:	Point on the circle
Point 3:	Orientation of the circle (optional)

The Circle (center, outside) command draws a circle based on a point at the center and a point on the outside of the circle. A third point can be used to specify the plane on which the circle lies.

Using the Command

Select the CIRCLE (CENTER, OUTSIDE) command. The program needs to know the form you want to save the circle in. The dialog box displays your three choices:

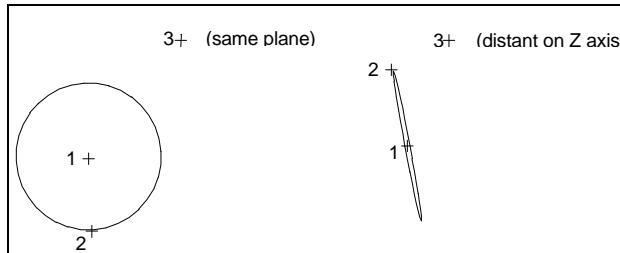
1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides; plane circles can be shaded.
3. **Line:** Stored as a line entity.



To choose an option click the button beside it. Normally, you'll want to save the circle in the CIRCLE form. If you want to use the circle as you would a solid, select PLANE. Save the circle as a LINE to make it possible to scale the circle or treat it as a line entity.

Example: Draw a circle using a specific point for the center.

Select the CIRCLE (CENTER, OUTSIDE) command and set a point near the center of the screen. Move the cursor away from the first point in any direction. Notice that a rubber-band representation of the circle is drawn using the cursor location as a point on the circle. Set a point when the circle is of the desired radius. Now, hold down **Ctrl+Shift** while moving the mouse forward or backward. This moves the cursor away from or toward you on the Z axis respectively. As you do this, the circle will pivot on the hinge set by the first two points. When the circle lies in the plane that you want it to, set the third point.



Circle (Center-Radius) Command

Menu:	DRAW
Submenu:	CIRCLES
Menu Command:	CIRCLE (CENTER-RADIUS)
Toolbox Icon:	
Point 1:	Center of the circle
Point 2:	Orientation of the circle (optional)
Point 3:	Orientation of the circle (optional)

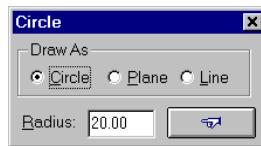
The Circle (Center-Radius) command draws a circle of a specified radius with a point set for the center. Two more points can be used to orient the circle in 3-D space.

Using the Command

Choose the CIRCLE (CENTER-RADIUS) command in the Main Toolbox. Enter the length of the radius in the RADIUS box. A rubber-band circle shows how the circle will be drawn. Set a point for the center of the circle. Press **Enter** or set a second point to orient the circle. Press **Enter** or set a third point to tilt the circle using the first two points as a hinge. A circle is drawn with the specified radius and with its center at the first point.

The circle can be saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the CIRCLE option is selected, the circle is saved in the drawing as a Circle entity. This is the way circles are normally saved with DesignCAD 3D MAX.

The circle can be saved as a plane so it can be shaded, subtracted, etc.

The circle can also be saved as a line. This makes it possible to scale the circle or treat it as a line entity with other commands.

Usually, you will want to save the circle as a Circle entity or as a Plane entity.

Radius

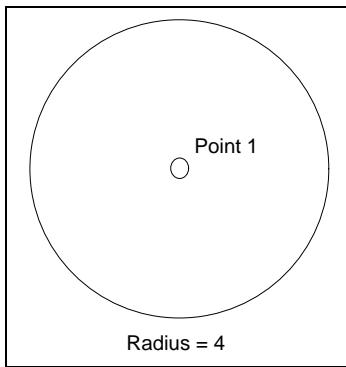
Enter the length of the radius of the circle in this box.

Same As

To have the radius length the same as another circle in the drawing, click the SAME AS button. Then set a point on the other circle in the drawing.

Example: Draw a circle with a radius of four.

Choose the CIRCLE (CENTER-RADIUS) command from the Main Toolbox. Enter **4** in the RADIUS box in the dialog box. When you return to the drawing, a rubber-band circle appears, showing how the circle will be drawn. Now set the point for the center of the circle. Press **Enter**. The circle is drawn with its center at the point and its radius four Drawing Units away.



Circle (Diameter) Command

Menu: DRAW
 Submenu: CIRCLES
 Menu Command: CIRCLE (DIAMETER)
 Toolbox Icon: 

Point 1: A point on the diameter of the circle
 Point 2: A point for the opposite end of the diameter
 Point 3: A point to align the circle in 3-D space (optional)

The Circle (Diameter) command draws a circle between two points, so that the two points lie on the diameter of the circle.

Using the Command

Set two points for the circle diameter. If necessary, you can set a third point to define the plane on which the circle lies. If you just want to set two points, press **Enter** after you have set them.

The circle is saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the **CIRCLE** option is selected, the circle is saved in the drawing as a Circle entity. This is the way circles are normally saved with DesignCAD 3D MAX.

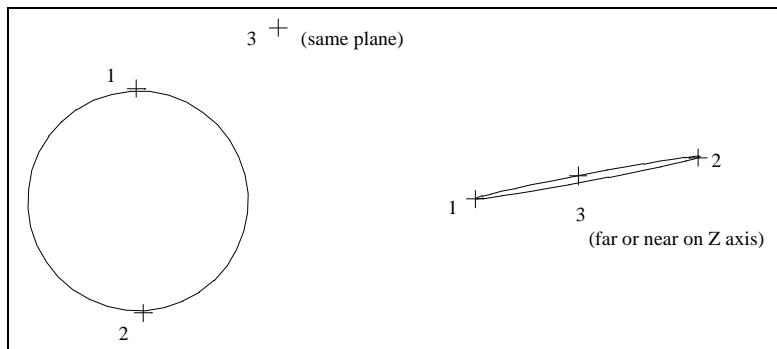
The circle can be saved as a plane so it can be shaded, subtracted, etc.

The circle can also be saved as a line. This makes it possible to scale the circle or treat it as a line entity with other commands.

Usually, you will want to save the circle as a Circle entity or as a Plane entity.

Example: Draw a circle using two points to define the diameter.

Select the CIRCLE (DIAMETER) command and set a point on the screen for a diameter point on the circle. As you move the cursor, a rubber-band circle is drawn to represent the circle, using the cursor position as the other diameter point. Set the second point when the circle is of the desired diameter. So far, the circle is in the XY plane. Hold down **Ctrl+Shift** while moving the mouse forward or backward and notice how the circle swings on a hinge set by the first two points. When the circle lies in the plane you want it to, set the third point.



Circle Tangent to Two Lines Command

Menu: DRAW

Submenu: CIRCLES

Menu Command: CIRCLE TANGENT TO TWO LINES



Toolbox Icon:

Point 1: First line tangent to circle

Point 2: Second line tangent to circle

The Circle Tangent to Two Lines command draws a circle of a specified radius which is tangent to two lines. The two lines must lie in the same plane unless you are currently in 2-D Mode.

Using the Command

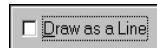
With two lines drawn, select the CIRCLE TANGENT TO TWO LINES command. Enter the radius of the circle in the RADIUS box in the dialog box. Set a point on each of the two lines which are to be tangent to the circle. A circle with the specified radius is drawn tangent to the two lines.



Note: If the two lines do not lie in the same plane, and the circle is drawn tangent to the two lines while in 2-D Mode, the circle is drawn at the Z coordinate the cursor was on before the switch to 2-D Mode. This will be apparent when you switch back to 3-D Mode.

The circle is saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the **CIRCLE** option is selected, the circle is saved in the drawing as a Circle entity. This is the way circles are normally saved with DesignCAD 3D MAX.

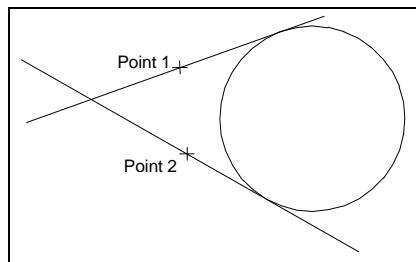
The circle can be saved as a plane so it can be shaded, subtracted, etc.

The circle can also be saved as a line. This makes it possible to scale the circle or treat it as a line entity with other commands.

Usually, you will want to save the circle as a Circle entity or as a Plane entity.

Example: Draw a circle with a radius of 10 tangent to two lines that lie on the same plane.

Select the **CIRCLE TANGENT TO TWO LINES** command and enter a radius of **10** in the **RADIUS** box in the dialog box. Set a point on one of the lines. When you move the cursor to the other line, a rubber-band circle will be drawn tangent to the two lines. Set the second point to insert the circle into your drawing.



Circle Tangent to Three Lines Command

Menu:	DRAW
Submenu:	CIRCLES
Menu Command:	CIRCLE TANGENT TO THREE LINES

Toolbox Icon:



Point 1: First line tangent to circle
 Point 2: Second line tangent to circle
 Point 3: Third line tangent to circle

The Circle Tangent to Three Lines command draws a circle which is tangent to three lines. The three lines must lie in the same plane unless you are currently in 2-D Mode.

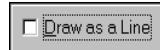
Using the Command

With three lines drawn, select the CIRCLE TANGENT TO THREE LINES command. Set a point on the first line. Set a point on the second line. After the second point is set, a rubber-band circle shows how the circle will be drawn. Set a point on the third line. A circle is drawn tangent to the three lines.

Note: If the three lines do not lie in the same plane, and the circle is drawn tangent to the three lines while in 2-D Mode, the circle is drawn at the Z coordinate the cursor was on before the switch to 2-D Mode. This will be apparent when you switch back to 3-D Mode.

The circle is saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the CIRCLE option is selected, the circle is saved in the drawing as a Circle entity. This is the way circles are normally saved with DesignCAD 3D MAX.

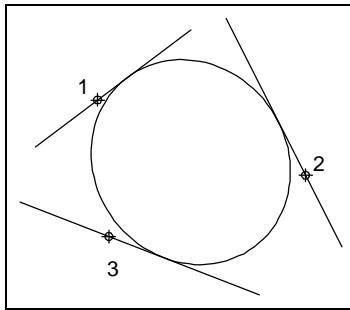
The circle can be saved as a plane so it can be shaded, subtracted, etc.

The circle can also be saved as a line. This makes it possible to scale the circle or treat it as a line entity with other commands.

Usually, you will want to save the circle as a Circle entity or as a Plane entity.

Example: Draw a circle tangent to three lines that lie on the same plane.

Select the CIRCLE TANGENT TO THREE LINES command. Set a point on one of the lines. Set a point on the second line. When you move the cursor to the third line, a rubber-band circle will be drawn tangent to the three lines. Set the third point to insert the circle into your drawing.



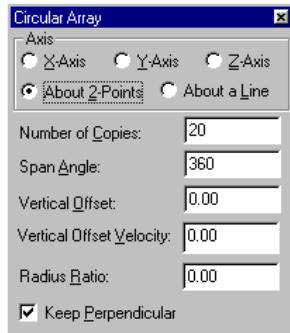
Circular Array Command

Menu: DRAW
 Menu Command: CIRCULAR ARRAY
 Toolbox Icon: 
 Point 1: Center of rotation
 Point 2: Axis of revolution (Used only with Two-Point Axis mode)

This command copies a selected object a specified number of times in a circular pattern.

Using the Command

You can set several options in the dialog box.



Axis

This is the axis of revolution. You can set the rotation along the X, Y, or Z axis, an axis defined by two points, or one defined by an existing line. Only the 2-Point option requires you to set a second point. With the line option you set a point on an existing line and the command uses that line as the axis of rotation. The other options will use your first point as the location of the axis.

Number of Copies

This is the total number of copies, including the original.

Span Angle

This is the angle in which the copies are to be placed. For example, you can use 360° to copy an object in a complete circle, such as the bolts on a wheel. You can use 180° to copy in a semicircle.

Vertical Offset

This is the distance along the axis of revolution between the first copy and the last copy. The offset can be used to copy objects in a spiral.

To make a spiral with more than one revolution, use a span angle of more than 360°. For example, to make a spiral with two complete revolutions, you can enter 720° for the span angle. Remember, the number of copies and the offset are the total for the entire operation, not just one revolution.

Vertical Offset Velocity

This is a ratio between the initial pitch of the offset and the final pitch of the offset. If a value of 5 is used, the final pitch will be 5 times greater than the initial pitch. If a value of 0 or 1 is used, the pitch will remain constant.

Radius Ratio

The distance from the selection handle to the axis is used for the array radius at the beginning of the array. The final radius for the array equals the beginning array radius multiplied by the Radius Ratio.

If a value of 1 is used for the Radius Ratio, the radius stays the same throughout the array. If a value greater than 1 is used, the distance from the axis to the duplicated items increases throughout the array. If a value less than 1 is used, the distance from the axis to the duplicated items decreases throughout the array.

Keep Perpendicular

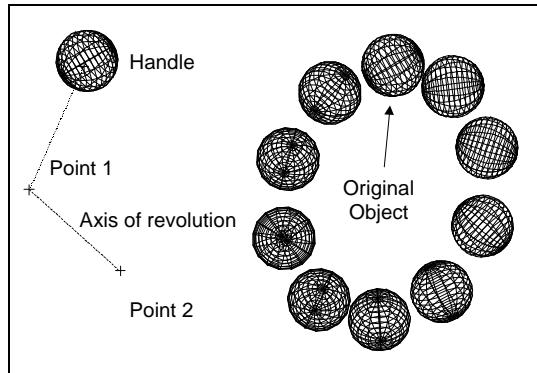
The Keep Perpendicular option is used to specify whether or not the copies for the array will rotate according to their angle of rotation in the array. If the option is checked, the copies will rotate with the array. If the option is not checked, the copies will still be copied in a circular pattern, but they will maintain the same orientation as the original.

Example: Copy an object 10 times in a circular pattern.

Select the object you want to copy. Next, choose the CIRCULAR ARRAY command. Enter the following settings in the appropriate fields in the dialog box:

NUMBER OF COPIES:	10
SPAN ANGLE:	360
VERTICAL OFFSET:	0

Select the TWO POINT option. Set a point a few inches below the object for the center of rotation and another, offset from the first, for the axis of revolution. It may help to think of this as swinging a ball on a string: The selection handle is where the string is connected to the ball, the center of rotation (Point 1) is where your hand holds the string, and the axis of revolution (Point 2) is the direction of your arm.



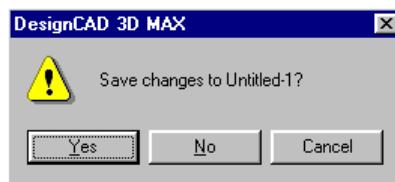
Close Command

Menu: FILE
 Menu Command: CLOSE

The Close command closes an open drawing.

Using the Command

Choose the Close command. If the drawing has changed, the program gives you the opportunity to save the changes:



Clicking YES saves the changes. Clicking NO closes the drawing, dropping all modifications since it was last saved. Clicking CANCEL or pressing **Esc** cancels the Close command and returns you to the drawing screen.

Close Digitizer Menu Command

Menu: TOOLS
 Submenu: DIGITIZER
 Menu Command: CLOSE DIGITIZER MENU
 Point 1: Point inside the menu area

The Close Digitizer Menu command closes the active digitizer menu.

Using the Command

Choose the CLOSE DIGITIZER MENU command. Set a point inside the menu area. The digitizer menu is closed.

See Also: [Add Menu Item Command](#), [Create Digitizer Menu Command](#), [Digitizer Tracing Mode Command](#), [Load Digitizer Menu Command](#), [Remove Menu Item Command](#), [Save Digitizer Menu Command](#)

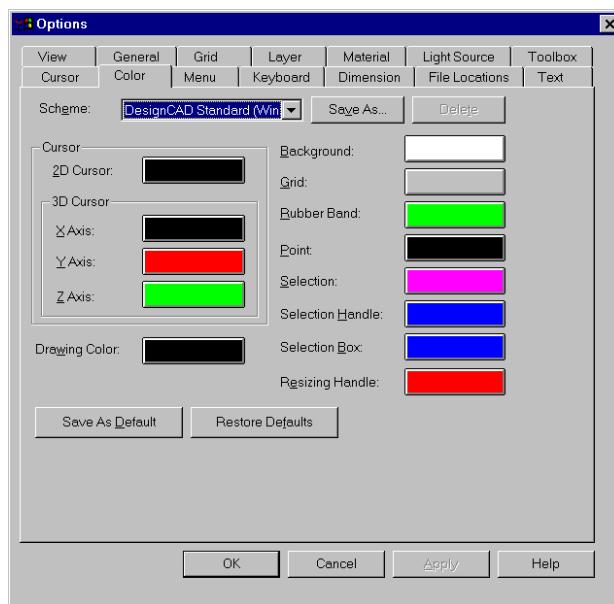
Color Options

Menu: OPTIONS
Menu Command: OPTIONS

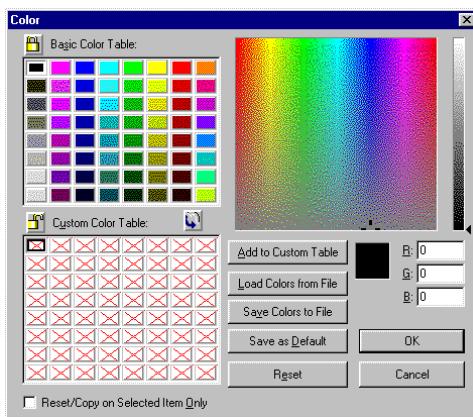
In the Color Options folder, you can set the color of the background, cursors, grid, points and various selection and rubber-band entities on the drawing screen.

Using the Command

Choose the OPTIONS command from the OPTIONS menu, and then click the COLOR tab to bring up the Color Options folder.



Click the button beside the item for which you want to change the color, and the Color Palette box appears.



Click to select the new color for the item. The selected color is highlighted with a black selection border around it. Click **OK** to set the color.

On the Color Options folder, the large button next to the item you are changing now shows the new color you selected. Click **APPLY** to make DesignCAD use this color and then **OK** to close the dialog box.

Hint: Before you click the **OK** button, click the **Save As Default** button if you want DesignCAD to use this color selection the next time you load the program.

Scheme

Select the color scheme to be used from the **SCHEME** list box.

To save a new color scheme, change the colors for the desired items. Click on the **SAVE AS** button. The **Save Scheme As** box appears. In the **SCHEME NAME** box enter the name of the file to save. This new color scheme may be selected from the **SCHEME** list box at any time until it is deleted. Any new custom color schemes will be listed at the bottom of the Scheme list.

To delete a custom color scheme, select the scheme name from the **SCHEME** list box. Click on the **DELETE** button. The scheme is removed from the Scheme list box.

Note: Only custom schemes you have created may be deleted.

2-D Cursor

This changes the color of the 2-D cursor.

3-D Cursor

X Axis

This changes the color of the X axis indicator line in the 3-D cross hairs cursor.

Y Axis

This changes the color of the Y axis indicator line in the 3-D cross hairs cursor.

Z Axis

This changes the color of the Z axis indicator line in the 3-D cross hairs cursor.

Drawing Color

This changes the color used as the default drawing color.

Background

This changes the color used as the background in the DesignCAD drawing windows.

Grid

This changes the color of the drawing grid. This option is the same as the Display Grid Color option in the Grid folder of the Options file box.

Rubber Band

This changes the color of rubber-band entities.

Point

This changes the color of DesignCAD's point indicators used during drawing commands.

Selection

This changes the color of a selection box.

Selection Handle

This changes the highlight color of a selection handle when a point or drawing entity is selected.

Selection Box

This changes the color of the selection box that appears when the cursor is used to drag an area around objects to be selected.

Resizing Handle

This changes the highlight color of resizing handles that appear around selections when the Use Resizing Handles command in the Options menu is enabled.

Restore Defaults

You can return to DesignCAD's original color settings by clicking the RESTORE DEFAULTS button.

Combine Lines Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	COMBINE LINES
Shortcut Key:	B

The Combine Lines command is used to merge connecting lines and arcs into a single line entity.

Using the Command

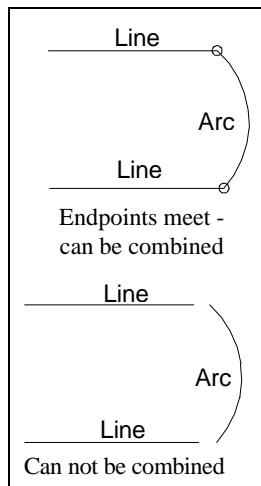
Select the entities you wish to combine into a single line. All selected entities **must** share endpoints to form a continuous line. That is, each line entity has to be connected to the next line. This command affects currently selected entities.

This command is similar to the Make Plane command, but the Make Plane command converts the selected lines into a plane instead of a single line entity.

Note: Since this command converts arcs into lines, any arcs will lose their center points.

Example: Convert two lines that meet at their endpoints into a single object.

Drag a selection box around the objects so that they are selected. Now, select the COMBINE LINES command. The lines are now combined into a single object. This can be seen easily when the new line is selected.



Command Dialog Command

Toolbox Icon:



The Command Dialog button is a toggle which displays or hides the dialog box for the current drawing command.

Using the Command

Select a drawing command. The Command Dialog button is pressed in, and if there are any options for that command the dialog box appears. Click on the Command Dialog button to release it. The dialog box for the drawing command is hidden and your view of the drawing screen is unobscured. If you decide that you need to set the options for the drawing command, click the Command Dialog button again and the dialog box will reappear.

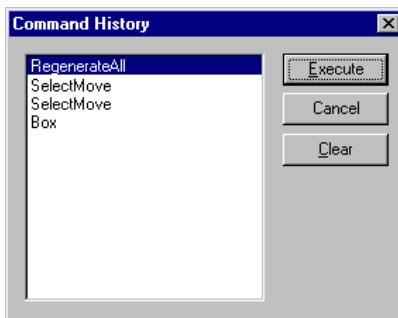
Command History Command

Menu:	TOOLS
Menu Command:	COMMAND HISTORY

The Command History command lists the commands that have been used in the current drawing session with the most recent first. Commands in the list may be executed using the Command History dialog box.

Using the Command

Select the COMMAND HISTORY command from the TOOLS menu. The Command History dialog box appears.



Choose a command from the list by clicking on it with the mouse. Click on the EXECUTE button to start the command. The Command History dialog box disappears and the execution of the command you chose begins.

To return to the DesignCAD drawing screen without starting a command, click the CANCEL button.

To clear the command history, click the CLEAR button.

Command Line Entry Command

Menu:	none
Shortcut Key:	Spacebar

Command Line Entry command lets you choose a command by typing the name of the command, or an alias, in the Command Line. With this method you don't have to select commands from the menu or by other means. Also, the program keeps a list of recently entered commands. Once you have entered a command, you can easily choose it again by pressing the Spacebar and using the up or down arrow keys to scroll to the command.

Using the Command

Press the **Spacebar**. The COMMAND box appears. Enter the name of the command or alias in the box and press **Enter**. If any options are associated with the command you have chosen, the

dialog box appears as usual. Press the **Tab** key to enter the dialog box and navigate to the option you want. Set the options and then press **Enter** to return to the drawing.

Note: Many of the commands let you enter the parameters for the command on the same line in the Command box.

Editing the Command File

The program associates command names and aliases with command IDs in the MACROCMD.INI file. When you enter a custom command name, the program still identifies it with the real command name. This gives you a great deal of flexibility in customizing commands. You can even have multiple aliases for the same command.

Commented lines begin with a semicolon. They are not allowed on the same line as an alias. Please use caution when editing the MACROCMD.INI file. Make a backup of the file before you try to edit it. Edit only the Add Custom Command section, part of which is shown below. If you edit other sections, you may cause commands to stop working correctly.

```
;=====ADD CUSTOM COMMAND NAME ENTRIES HERE=====
```

```
;
```

```
;These sample commands are laid out following the menu's structure.
```

```
;
```

```
; File Menu
```

```
;
```

```
Nu=ID_DCAD_FILE_NEW
```

```
Ld=ID_DCAD_FILE_OPEN
```

```
QF=ID_DCAD_FILE_CLOSE
```

```
Sv=ID_SAVEFILE
```

```
SvA=ID_SAVEFILE_AS
```

```
SvS=ID_SELECT_SAVE
```

Creating an Alias

First, create a blank line for the new alias. Then enter the alias you want to use followed by the equals sign (without spaces) and the actual name of the DesignCAD command. It is not necessary to comment out the existing alias.

Suppose you want to add Clo as an alias for the File Close command. Go to the end of the following line:

```
QF=ID_DCAD_FILE_CLOSE
```

Press **Enter** to start a new line. Now enter the new alias as shown below:

```
clo=ID_DCAD_FILE_CLOSE
```

You may, of course, copy the DesignCAD command name and paste it in instead of typing it yourself. Save the file when you are sure the information is correct. The next time you start DesignCAD 3D MAX, it reads the file you have edited. Then you can press the Spacebar and enter **clo** in the COMMAND box to close a file.

Example: Draw a line from the origin (0,0) to (10,10).

Press the **Spacebar** to activate the Command Line. Start the **LINE** command by entering **Line**. Then press the **Enter** key. Next, press the **Spacebar** again. In the **COMMAND** box enter **pointxyz 0,0,0** and press **Enter** to set the first point. Press the **Spacebar** a third time. The previous entry (**pxyz 0,0,0**) remains in the **COMMAND** box. Now change the **0,0,0** to **10,10,0** and press **Enter**. That sets the second point. Press **Enter** again to end the Line command. The line is drawn between those two points.

Cone Command

Menu: SOLIDS

Menu Command: CONE



Toolbox Icon:

Point 1: Center of the base of the cone

Point 2: Edge of the base of the cone

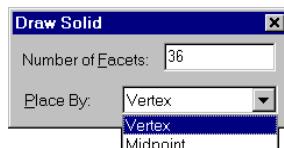
Point 3: Height and direction of the cone

The Cone command draws a cone.

Using the Command

Set a point for the center of the base of the cone, a second point at the edge of the base, and a third point for the cone height.

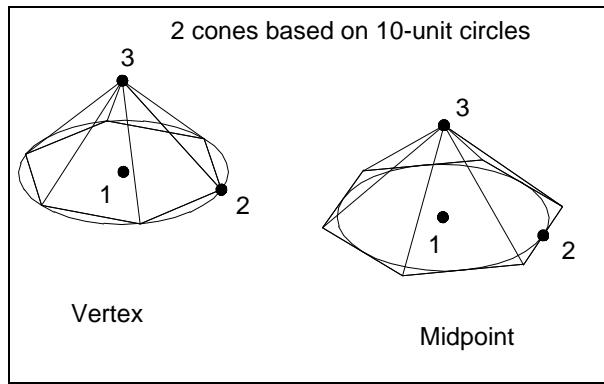
You can specify the number of sides or facets around the cone in the **NUMBER OF FACETS:** field in the dialog box.



You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Point 2. If you choose **VERTEX**, the base of the cone is inscribed by a circle of that radius. If you choose **MIDPOINT**, the base of the cone circumscribes a circle of that radius. This is normally not significant, but it can be important for some precision drawings.

Example: Draw a cone.

Select the **CONE** command. Next set a point for the center of the base. Move the cursor out along the Y axis and set the second point for the radius of the cone. Now move the cursor up until the cone is the desired height and set the third point. The cone is inserted into the drawing.



See Also: [Truncated Cone Command](#)

Continue Recording Command

Menu:	TOOLS
Menu Command:	CONTINUE RECORDING

The Continue Recording command restarts the recording of a macro. After suspending the recording of a macro, click the CONTINUE RECORDING button to resume recording.

Using the Command

After stopping or pausing a macro recording, choose the CONTINUE RECORDING command. The macro resumes the recording.

See Also: [Macro Record Command](#), [Stop Recording Command](#), [Pause Recording Command](#)

Control Panel Command

Menu:	ANIMATION
Menu Command:	CONTROL PANEL

The Control Panel command is a toggle that hides or shows the Animation Control Panel depending on the panel's current status.

Using the Command

While in Animation Mode, select the CONTROL PANEL command from the ANIMATION menu. If the Animation Control Panel was visible, it will be hidden. If the Control Panel was hidden, it will be made visible.



See Also: [Animation Mode Command](#), [Time Line Command](#)

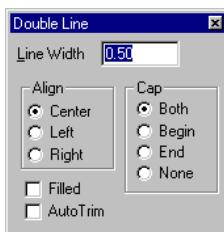
Convert to Double Line Command

Menu: EDIT
 Menu Command: CONVERT TO DOUBLE LINE
 Point 1: Entity to be converted to a double entity

The Convert to Double Line command converts existing 2-D entities into double entities. Entities that are converted using Convert to Double Line are automatically paralleled a specified distance and direction from the original. The ends of the entity may be capped or left open independently, and the double entity may be filled and/or automatically trimmed. Double lines simplify several common types of drawings, including house plans and land plots.

Using the Command

Choose the CONVERT TO DOUBLE LINE command from the EDIT menu. The Double Line dialog box appears.

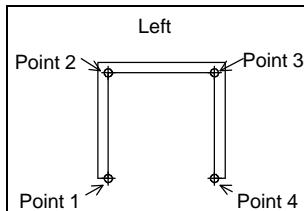


Line Width

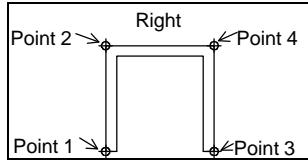
Sets the width of the entity (the amount of space between the two parallel entities) in drawing units.

Align

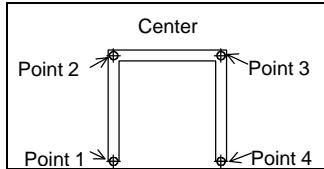
Select Center, Left, or Right to set the position of the width of the double entity. If Left is selected, the width of the entity will extend to what would be your left if you were standing on the first point set for the entity facing the second point set for the entity.



If Right is selected, the width of the entity will extend to what would be your right if you were standing on the first point set for the entity, facing the second point of the entity.



If Center is selected, the width of the entity will extend an equal distance on both sides of the points.



Note: If the double entity is converted with the Left or Right Align option selected, you will be able to snap to both sides of the finished double entity. If the double entity is converted with the Center option selected, you will be able to snap to both sides and the center of the double entity.

Cap

Select Both, Begin, End, or None to specify which ends of the double entity are capped or left open.

Filled

Check the Filled option for a filled entity, or leave it unchecked for a hollow entity.

Auto Trim

If the Auto Trim option is checked, the double entity will automatically be trimmed where it overlaps itself or another double entity.

Note: The Explode command can be used to explode a double entity into vectors. If you are going to cross one double entity over another, explode the existing double entity and uncheck the Auto Trim box before drawing the second double entity to ensure that all four entities will be visible in the intersection.

Set a point on the simple 2-D entity (such as arc, circle, curve, or line) to be converted to a double entity. The entity is converted to a double entity with the options you chose.

See Also: *Double Line Mode Command*

Copy Command

Menu:	EDIT
Menu Command:	COPY
Shortcut Key:	Ctrl+C
Toolbox Icon:	

The Copy command copies selected objects from the DesignCAD 3D MAX drawing screen to the Windows Clipboard, *leaving the original objects still in the drawing*. From the Clipboard, they can be pasted back into DesignCAD 3D MAX or into other Windows applications.

Using the Command

Select the object or objects to be copied. Choose the COPY command. The program copies the selected items to the Clipboard.

Example: Copy an object from your drawing to the Windows Clipboard.

Select the object and click the COPY icon. Then select PASTE from the EDIT menu. Drag the green box to the location where you want to put the copy and click the left mouse button. The object is inserted into the drawing.

If you want to paste the object into another drawing or Windows application, you can do so without recopying. The object remains in the Clipboard until another object is cut or copied to the Clipboard, replacing it.

See Also: ***Copy Image Command, Cut Command, Paste Command***

Copy Image Command

Menu:	EDIT
Menu Command:	COPY IMAGE
Shortcut Key:	Ctrl+Shift+C
Point 1:	First corner of rubber-band box
Point 2:	Opposite corner of the box

The Copy Image command copies text or graphics onto the Clipboard. Copying text or graphics to the Clipboard replaces the contents previously stored there.

The Copy Image command is similar to the Copy command except the entities are not selected prior to selecting the command. The Copy Image command copies only what is enclosed in the rubber-band box. The image is treated like a Metafile because the OLE object information is bypassed.

Using the Command

Choose the COPY IMAGE command from the EDIT menu. Set a point for the first corner of a rubber-band box that will be used to define the image to be copied to the clipboard. Use the mouse to enclose the image with the rubber-band box. When the rubber-band box surrounds the image you want copied, set another point for the opposite corner of the rubber-band box. A copy of the image is placed onto the Clipboard.

Hint: You can also use the Copy Image command to copy **part** of several entities onto the Clipboard. The image is treated like a drawing instead of a CAD object.

See Also: ***Copy Command, Cut Command, Paste Command***

Create Digitizer Menu Command

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	CREATE DIGITIZER MENU

The Create Digitizer Menu command creates a digitizer menu. The menu created with this command does not appear on the screen. The menu contains several selection boxes. These boxes are spaces where commands are selected.

Using the Command

To create a digitizer menu, follow these steps:

1. Draw the menu template. This is the paper template that will be affixed to the digitizer. To start with, draw a border box the size of the menu. The menu can be any size, but the larger the menu template is, the more of the digitizer drawing area the menu will take up.
2. On the template, draw individual "command" boxes for each DesignCAD command you want on the menu. These command boxes can be any size you want. Draw text or a picture to be associated with each command in the command box.
3. Print the finished template at a scale of 1. Cut it out and affix it to the digitizer, inside the active drawing area of the digitizer.
4. Choose CREATE DIGITIZER MENU from the DIGITIZER submenu in the TOOLS menu. Choose the method of creation in the Digitizer Menu dialog box. Then click the OK button.
5. Now set a point in the lower-left corner of the digitizer template. Set a point in the upper-right corner of the digitizer template.
6. Next, set a point in the lower-left corner of the command box. Set a point in the upper-right corner of the command box.
7. Enter the command name in the COMMAND box. The DesignCAD macro command names can be found in the DesignCAD Macro Command Name section of the MACROCMD.INI file. The command names appear to the right of the equal mark in the MACROCMD.INI file.
8. Set two more points for the next command box, or press the **Enter** button to end the command.

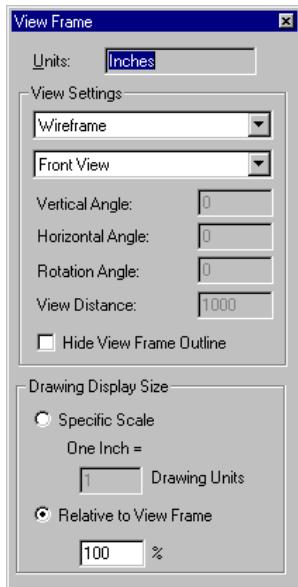
Create View Frame Command

Menu:	VIEW FRAME
Menu Command:	CREATE VIEW FRAME
Toolbox Icon:	

The Create View Frame command is used to create the View Frames to be printed in a Paper Space. A View Frame is drawn the same way that a selection rectangle is drawn.

Using the Command

Make sure DesignCAD is in Paper Space Mode. Choose the CREATE VIEW FRAME command from the VIEW FRAME menu or click on the Create View Frame icon in the Paper Space dialog box. The View Frame dialog box appears.



Select the View Settings and Drawing Display Size for the View Frame.

Now move the cursor to the location in the Paper Space at which you want to place the first corner of the View Frame. Press and hold the mouse button. Drag the mouse until the rubber-band preview is the size you want the View Frame. Release the mouse button. The rubber-band preview is replaced with a View Frame border with your drawing inside.

The View Frames can also be drawn using the Point Polar and Point Relative commands.

Note: The Create View Frame command is only available when DesignCAD is in Paper Space Mode. The View Frame menu appears in the Command Menu when Paper Space Mode is enabled.

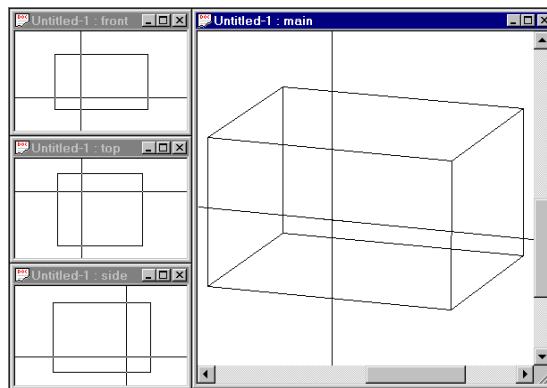
Crosshair Command

Menu:	OPTIONS
Menu Command:	CROSSHAIR
Shortcut Key:	F5

The Crosshair command indicates the cursor's position with cross hairs in all views.

Using the Command

Using the Crosshair command helps you easily distinguish the cursor's position, in relation to objects, from several angles. This makes it easy to see the cursor's location in all three dimensions. When you choose the command, the cross hairs become visible on the drawing screen. To remove the cross hairs, choose the CROSSHAIR command.



Cursor Command

Menu:	OPTIONS
Menu Command:	CURSOR
Shortcut Key:	I

The Cursor command changes the distance the cursor moves when you use the cursor movement keys on the keyboard.

Using the Command

Choose the CURSOR command. The Cursor Options folder appears. Set the options to meet your drawing needs. (See "Cursor Options" for details.)

Example: Set the large cursor step at one foot and the small cursor step at one inch.

For this example, we must assume that one drawing unit represents one foot. First, choose the CURSOR command. Then click the radio button for RELATIVE TO DRAWING. Enter **1** for the LARGE STEP SIZE and **1/12** or **1"** for the SMALL STEP SIZE.

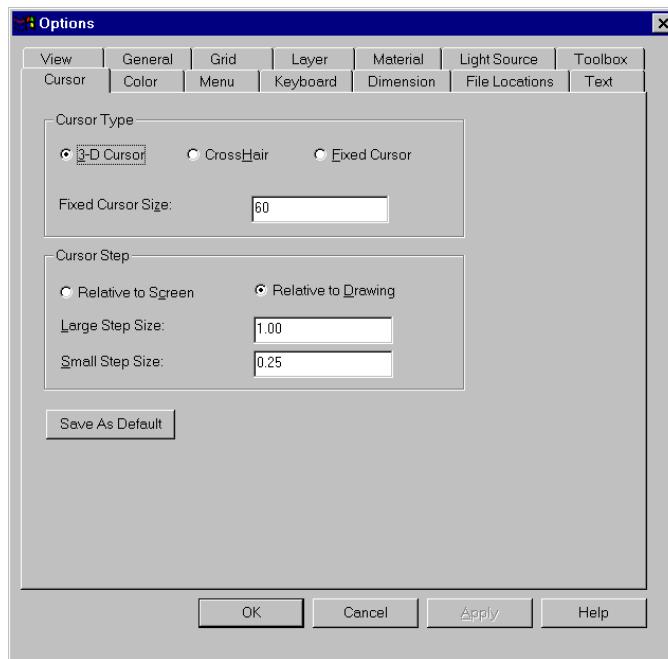
Cursor Options

Menu:	OPTIONS
Menu Command:	OPTIONS

In the Cursor Options folder, you can set the size of the cursor and the small and large step sizes for it.

Using the Command

Choose the OPTIONS command, and then click the CURSOR tab to bring up the Cursor Options folder.



Cursor Type

3-D Cursor

This causes the cursor to appear closer or further away according to its location on the Z axis.

Crosshair

Choosing this option turns the cursor into a pair of cross hairs that extend to the ends of the screen.

Fixed Cursor

Choosing this option causes the cursor to remain one size during the drawing session.

Fixed Cursor Size

Choosing this option sets the size of the cursor in Drawing Units.

Cursor Step

Relative to Screen

With this option selected, the cursor will always move the same number of pixels across the screen, regardless of the zoom factor.

Relative to Drawing

This option makes the cursor always move the same number of Drawing Units, regardless of the zoom factor.

Large Step Size

Choosing this option sets the number of Drawing Units that the cursor moves when you press the **Arrow** keys or **Ctrl+Home** or **Ctrl+End**.

Small Step Size

Choosing this option sets the number of Drawing Units the cursor moves when you press **Shift** while using the **Arrow** keys or **Ctrl+Home** or **Ctrl+End**.

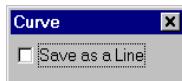
Curve Command

Menu: DRAW
 Submenu: LINES
 Menu Command: CURVE
 Shortcut Key: C
 Toolbox Icon: 

Point 1-n: Points for the curve to pass through

The Curve command draws a cubic spline curve through three or more points. The points for a curve do not have to lie in the same plane.

The curve can be saved as a line by clicking on the **SAVE AS A LINE** option in the dialog box.

**Example: Draw a curve.**

Select the CURVE command. Set three points on the screen. A curve is drawn through the three points.

Hint: To make a "corner" in the curve, set two consecutive points in the same location. This will cause two separate curves to be drawn, one on each side of the corner.

Curve to Line Command

Menu: EDIT
 Submenu: SELECTION EDIT
 Menu Command: CURVE TO LINE

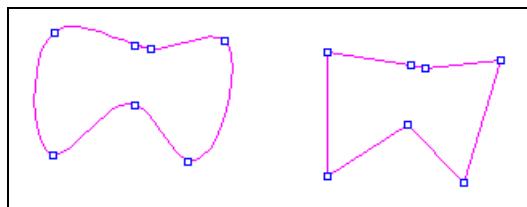
The Curve to Line command changes a curve entity into a line using the same points as those that defined the curve.

Using the Command

Select the curve that you wish to convert to a line. Choose the CURVE TO LINE command. The selected curve is changed to a line.

Example: Change a closed curve into a closed line.

Select the curve to be changed. Choose the CURVE TO LINE command. The curve becomes a line with the same number of points.



See Also: [Line to Curve Command](#)

Custom Color Command

Menu:	OPTIONS
Menu Command:	CUSTOM COLOR
Toolbox Icon:	

The Custom Color command is used to edit the currently selected drawing color, the Basic Color Table, and the Custom Color Table.

Using the Command

Choose the CUSTOM COLOR command. Move the Color Selector to change the color in the Color | Solid box. Moving left or right will change the hue of the color and moving up or down will change the saturation of the color. The brightness of the color can be adjusted by moving the arrow beside the brightness bar. A custom color can also be set by entering new values in the Red, Blue and Green boxes.

Add to Custom Table

Click the ADD TO CUSTOM TABLE button to put the color in the selected cell in the Custom Color Table.

Swapping Tables

To lock or unlock the Basic Color Table or the Custom Color table, click on its lock icon

To move colors in the Basic Color Table to the Custom Color table, make sure that the Basic Color Table is locked , the Custom Color Table is unlocked , and click the SWAP button

To move colors in the Custom Color Table to the Basic Color table, make sure that the Basic Color Table is unlocked , the Custom Color Table is locked , and click the SWAP button .

To move colors in the Basic Color Table to the Custom Color table and colors in the Custom Color Table to the Basic Color Table, make sure that both tables are unlocked  and click the SWAP button .

Note: Any time the Basic Color Table is changed, the corresponding drawers in the Color Toolbox will also change.

Load Colors From File

Click the LOAD COLORS FROM FILE button to load a custom color file that has been saved using SAVE COLORS TO FILE.

Save Colors to File

Click the SAVE COLORS TO FILE button to save the current Basic Color Table and Custom Color Table to a file that can be opened using LOAD COLORS FROM FILE.

Save as Default

Click the SAVE AS DEFAULT button to use the current Basic Color Table and Custom Color Table in new drawings and subsequent DesignCAD drawing sessions.

Reset

Click the RESET button to restore the original Basic Color Table and Custom Color Table.

Reset/Copy on Selected Item Only

If this option is checked, only the selected color cells will be reset or copied when a function is performed.

When the selected color is the one you want to assign as the current drawing color, click OK or press **Enter**. That color appears in the Main Toolbox as the current color, and new objects will be drawn in that color.

Example: Change the color used to draw entities.

Select the CUSTOM COLOR command. Move the Color Selector to change the color in the Color | Solid box. When you have set the RGB values, click OK.

Custom Hatch Pattern Load Command

Menu:	FILE
Submenu:	CUSTOM HATCH PATTERN
Menu Command:	LOAD

This command loads previously saved custom hatch patterns and makes them available for use in DesignCAD.

Using the Command

Choose the CUSTOM HATCH PATTERN LOAD command. The LOAD HATCH PATTERN box appears. In the FILE NAME box enter the name of the pattern file (*.dch) to load, and in the Look In box select the location of the pattern file. Then click OPEN.

Any custom hatch patterns contained in the pattern file are loaded and become available in DesignCAD's hatch commands. The custom hatch patterns appear in the hatch commands with "AD_" appended to the beginning of the original hatch pattern name.

See Also: *Custom Hatch Pattern Save Command*

Custom Hatch Pattern Save Command

Menu:	FILE
Submenu:	CUSTOM HATCH PATTERN
Menu Command:	SAVE

This command saves hatch patterns contained in imported drawings.

Using the Command

Import the drawing that contains the hatch pattern(s) you want to save. Choose the CUSTOM HATCH PATTERN SAVE command. The Save Hatch Pattern box appears. In the FILE NAME box enter the name for the new file that will contain the custom hatch pattern(s). In the STORE IN box select the location where you want to store the file. Then click SAVE. The file is saved. The custom hatch patterns contained in the new file are saved with "AD_" appended to the beginning of the original hatch pattern name.

See Also: *Custom Hatch Pattern Load Command*

Cut Command

Menu:	EDIT
Menu Command:	CUT
Shortcut Key:	Ctrl+X
Toolbox Icon:	

The Cut command removes a selected object or group of objects from the drawing screen and places them on the Windows Clipboard. Once the selected object or objects are on the Windows Clipboard, you can paste the contents back into DesignCAD 3D MAX, DesignCAD 3000, DesignCAD 2000, DesignCAD 97, DesignCAD 3D, DesignCAD 2D, and many other Windows applications.

Using the Command

Select the object or objects you want to remove from the drawing and place in the Clipboard. Then choose the CUT command. The selection is cut to the Clipboard.

Example: Remove an object from your drawing and place it on the Windows Clipboard.

Select the object and click the CUT icon. The object is removed from the drawing. Then select PASTE from the EDIT menu. Drag the green box to the location where you want to replace the object and click the left mouse button. The object is then inserted into the drawing.

See Also: *Copy Command, Paste Command*

Cut Corner Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	CUT CORNER
Point 1:	Corner to be cut

The Cut Corner command cuts a portion off of the corner of a solid and replaces the corner with a flat face at a specified depth.

Using the Command

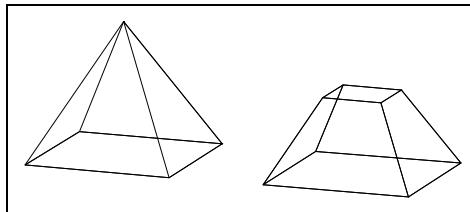
Choose the CUT CORNER command. In the RADIUS: box in the dialog box, enter the depth—the distance from the corner along all of the edges that form the corner—to the flat surface.



Next, set a point on the corner. The corner is cut off to the specified depth, or radius.

Example: Cut a corner off of a pyramid.

Choose the CUT CORNER command and set the RADIUS: in the dialog box. Set a point on one of the corners of the pyramid. The corner of the pyramid is cut off to the specified depth.



Cut Edge Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	CUT EDGE
Point 1:	Edge to be cut

The Cut Edge command cuts a portion off of the edge of a solid and replaces the edge with a flat face at a specified depth.

Using the Command

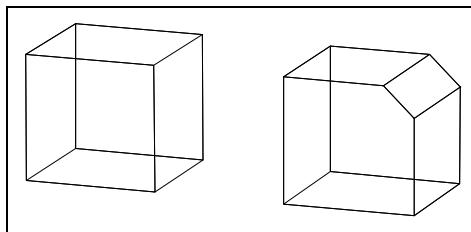
Choose the CUT EDGE command. In the RADIUS: box in the dialog box, enter the depth—the distance from the edge along the two surfaces that form the edge—to the flat surface.



Next, set a point on the edge. The edge is cut off to the specified depth, or radius.

Example: Cut an edge off of a box.

Choose the CUT EDGE command and set the RADIUS: in the dialog box. Set a point on one of the edges of the box. The edge of the box is cut off to the specified depth.



Cut Plane Command

Menu:	EDIT
Menu Command:	CUT PLANE
Point 1:	Cutting line
Point 2:	Plane to be cut

The Cut Plane command can be used to slice a plane into two or more pieces along a line.

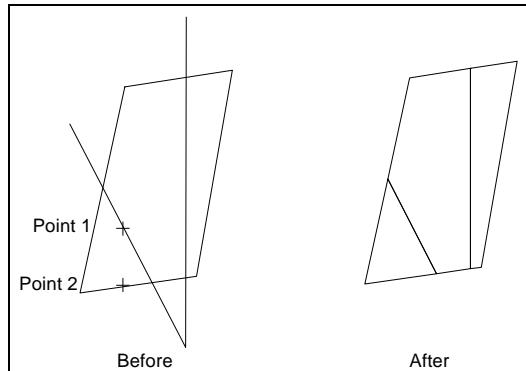
Note: The cutting line, or part of it, is erased during this command, so make a copy if necessary. If you forget, you can Undo the command, copy the line, and re-cut the plane.

Using the Command

The Cut Plane command requires a plane to cut and a line to cut it with. Choose the CUT PLANE command. Set a point on the cutting line, first, and then a point on the plane to be cut.

Example: Divide a plane along a line which spans it.

Choose the CUT PLANE command. Set one point on the line and another on the plane. The line disappears, and the plane is now divided into sections.



Cylinder Command

Menu: SOLIDS

Menu Command: CYLINDER

Toolbox Icon:



Point 1: Center of the cylinder face

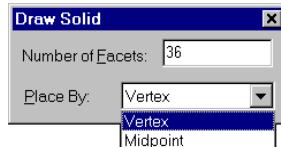
Point 2: Radius of the cylinder

Point 3: Length and direction of the cylinder

The Cylinder command draws a solid cylinder.

Using the Command

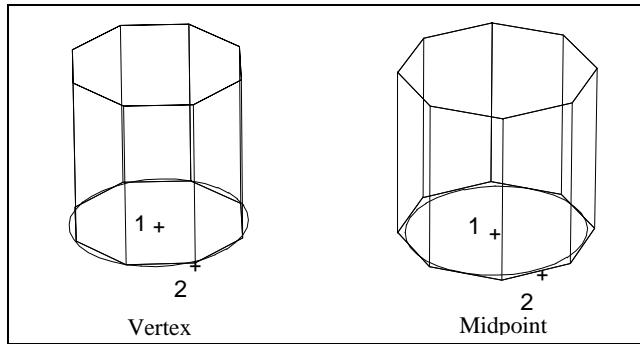
You can specify the number of sides or facets around the cylinder in the NUMBER OF FACETS box in the dialog box.



You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Point 2. If you choose VERTEX, the base of the cylinder is inscribed by a circle of that radius. If you choose MIDPOINT, the base of the cylinder circumscribes a circle of that radius. This is normally not significant, but it can be important for some precision drawings.

Example: Draw a cylinder in your drawing.

Select the CYLINDER command. Set a point for the center of the base. Move the cursor out along the Y axis and set the second point for the radius of the cylinder. Next, move the cursor up until the cylinder is of the desired height, and set the third point. The cylinder will be inserted into the drawing.



DesignCAD Help Command

Menu:	HELP
Menu Command:	DESIGNCAD HELP
Shortcut Key:	F1

To get help quickly as you work, choose the DesignCAD Help command. When the Help window appears, you can display information on DesignCAD commands, or get a definition for a term you do not understand. You can also get help whenever you see an error message or if you need more information about an option in a dialog box.

To get information quickly about a command or task, use on-line Help. You can display instructions, then complete your tasks without having to turn away from the screen.

With Help, you can:

- Search for topics associated with a word or phrase.
- View instructions in Help while you perform a task in DesignCAD.
- Add your own notes to a Help topic.
- Mark topics you often use so you can jump to them quickly.
- Print a copy of a Help topic.

Using the Command

Press **F1** to: display the Contents topic for DesignCAD 3D MAX Help; get help on a selected command; find out more about the options in the current dialog box; or see an explanation of an error message.

You can view Help side-by-side with a drawing so that both windows remain fully visible while you work. To display Help and your drawing at the same time, resize the windows so they do not overlap.

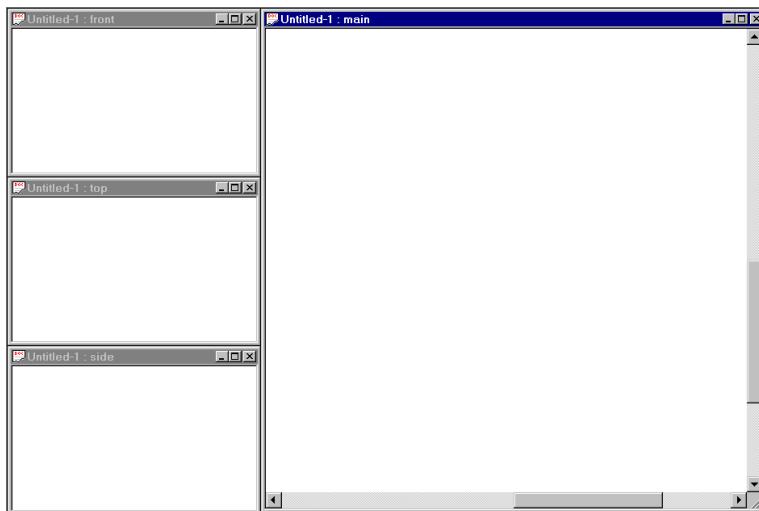
DesignCAD Tile Command

Menu:	WINDOW
Menu Command:	DESIGNCAD TILE

The DesignCAD Tile command arranges the active drawing windows according to the DesignCAD Tile setting. The default setting places the Perspective view in the large window on the right side of the screen, and the Front, Top, and Side views stacked vertically on the left. This is a convenient window arrangement for working in 3-D space, when using 3-D Selection Mode, and when working in 2-D or 3-D Mode with a large object which contains small items that need to be zoomed for detailed drawing.

Using the Command

Choose the DESIGNCAD TILE command in the WINDOW menu. The program arranges the windows according to the DesignCAD Tile setting which can be changed with the Set As DesignCAD Tile command. The default setting has Perspective, Front, Top, and Side views.



See Also: *Restore DesignCAD Tile Command, Set As DesignCAD Tile Command*

Digitizer Tracing Mode

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	DIGITIZER TRACING MODE

The Digitizer Tracing Mode command lets you zoom the screen image of the drawing you are tracing while retaining the present scale on the digitizer pad.

Using the Command

Make sure there is a check mark next to the DIGITIZER TRACING MODE indicating that it's active. Use the Zoom commands to zoom the current screen image of your drawing. The Zoom commands will not change the relationship between the drawing information being entered and the digitizer pad. Zoom commands only change the size or location of the drawing on the screen.

See Also: *Zoom Command, Zoom In Command, Zoom Out Command*

Dimension Command

Menu: DIMENSION

Menu Command: DIMENSION

Shortcut Key: @



Toolbox Icon:

Point 1: Start of distance to be measured

Point 2: End of distance to be measured

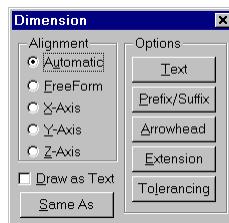
Point 3: Dimension text location

The Dimension command places dimensions in your drawing.

Using the Command

Choose the DIMENSION command. To add a dimension to your drawing, set two points for the distance to be measured. Then set a third point for the dimension text location.

Several options are available in the dialog box.



Automatic

- Selecting the AUTOMATIC option causes the program to determine automatically which axis you wish to measure based on where you place the dimension text.

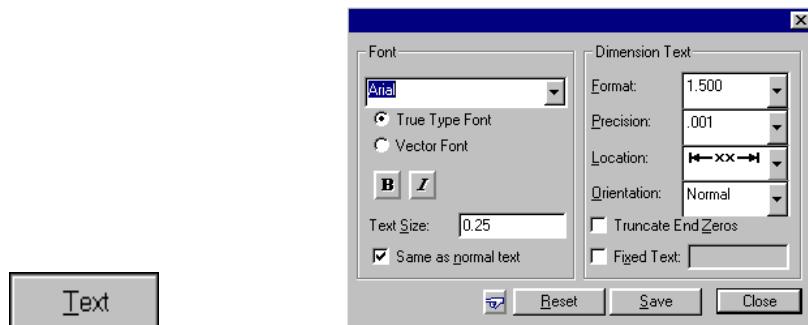
Free Form

- Selecting the FREEFORM option lets you measure along any arbitrary direction. The dimension text will always be placed parallel to the distance measured.

X-Axis, Y-Axis, Z-Axis

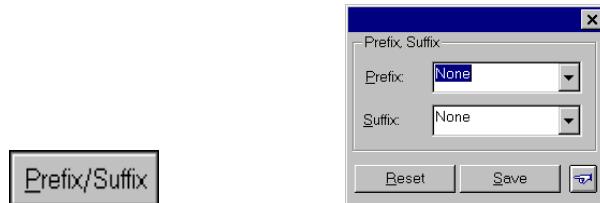
- Selecting any of these causes DesignCAD to measure only the distance parallel to that axis along the measured path.

Text



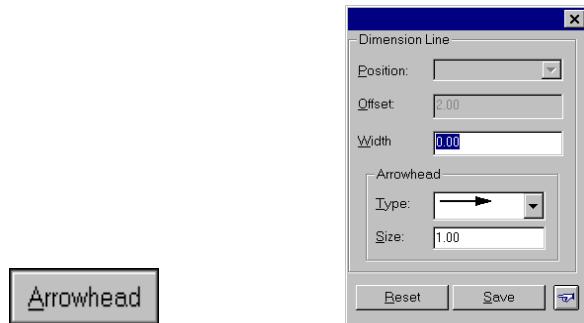
- Any Windows font can be used for dimensions. The dimension text size can be specified in Drawing Units.
- The Format option offers several different numeric formats for the dimensions. The **Shift+F12** keys may be used to toggle through the dimension formats without opening the Dimension Text dialog box.
- Specify the number of digits to the right of the decimal point with the PRECISION option.
- The dimension text can be placed inside or outside the extension line, to the right or left of the dimension arrows, or between the terminators. The **F12** key may be used to toggle through the text locations without opening the Dimension Text dialog box.
- The dimension text orientation may also be changed. The **Ctrl+F12** keys may be used to toggle through the text orientations without opening the Dimension Text dialog box.

Prefix/Suffix

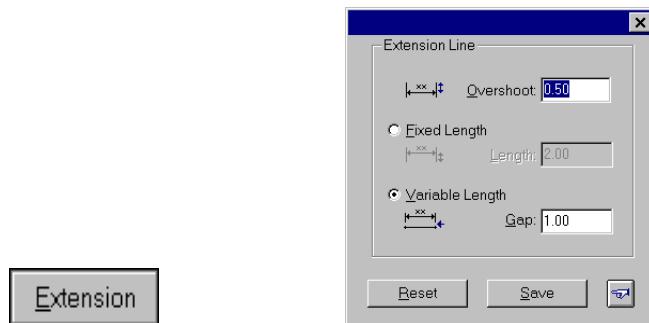


- The PREFIX option lets you add a notation before the dimension measurement. You can keep a list of up to five custom notations that are available for dimensioning.
- The SUFFIX option lets you add a notation after the dimension measurement. You can keep a list of up to five custom notations that are available for dimensioning.

Both boxes work the same. Click on the down arrow, choose the custom number, and enter the notation you want to add to your drawing.

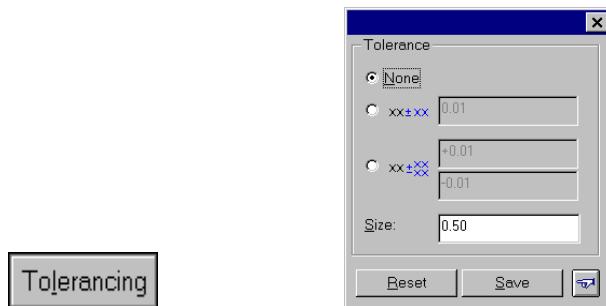
Arrowhead

- The dimension line can be positioned inside or outside the extension line.
- Enter the amount of offset in the OFFSET box.
- In the TYPE and SIZE boxes specify the arrowhead options you want to use.
- In any dimension option box, click the RESET button to reset the options to their usual settings.
- In any dimension option box, click the SAME AS button to copy dimension settings from a dimension in the drawing.

Extension

- Specify the amount of the overshoot in the OVERSHOOT box.
- You can choose whether you want FIXED LENGTH or VARIABLE LENGTH.
- Specify the amount of the gap in the GAP box.

Tolerancing



- The dimension tolerance is the positive and negative values showing the amount of variation allowed from the standard measurement.
- The NONE option allows no tolerance for the dimension, meaning that it must be exact.
- The second option allows you to set one value that will be used as the positive and negative dimension tolerance.
- The third option allows you to set separate values for the positive dimension tolerance and the negative dimension tolerance.

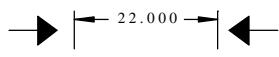
Draw as Text

- The default setting (Draw as Dimension) causes DesignCAD to update the dimension text anytime the dimension changes size (e.g., when resetting Drawing Units, or stretching an object and its associated dimension). Checking the DRAW AS TEXT box forces the program to keep the same dimension, regardless of changes.

Same As

- Clicking on the SAME AS button lets you use the same options as another dimension in your drawing. After choosing Same As, set a point on the dimension in the drawing. The new dimension will have the same settings.

Extension line



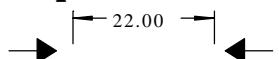
The extension line is perpendicular to the dimension line and is drawn at the end of the dimension line.

Overshoot



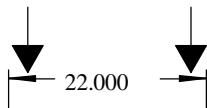
The extension line overshoot length is the distance the extension line extends past the dimension line.

Gap



The extension line gap size is the distance between the points set for dimensioning and the beginning of the extension line.

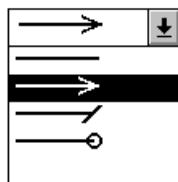
Terminator or arrowhead



The terminator, or arrowhead, is the entity drawn at the end of the dimension line.

Arrowhead styles

You can select the arrow style you want by pulling down the Arrow Type list box in the Dimension Line dialog box.



Several arrow styles are available.

Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

Example: Measure the distance between two points in your drawing and insert that figure as a dimension notation.

Choose the DIMENSION command. Set a point on the first point to be dimensioned and another on the second. As you move the cursor away from the points, a rubber-band dimension line and text box appear on the screen. When the dimension is in the desired location, set the final point to insert the dimension in your drawing.

Dimension Angle Command

Menu:	DIMENSION
Submenu:	DIMENSION ANGLE
Toolbox Icon:	
Point 1:	Center of the angle to be measured
Point 2:	Beginning of the angle
Point 3:	Endpoint of the angle
Point 4:	Distance at which the dimension text is to be inserted

The Dimension Angle command lets you add the dimensions of angles to your drawing.

Using the Command

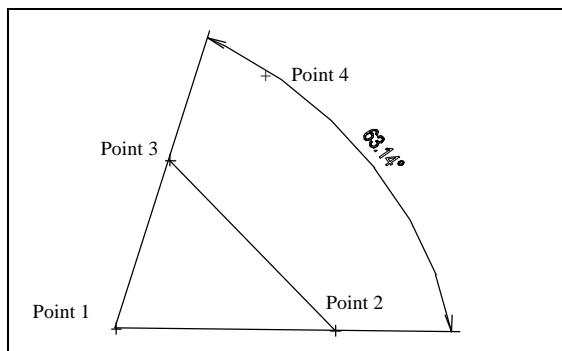
Choose the Dimension Angle command. Set a point for the center of the angle to be measured. Then set points for the beginning and end of the angle. Finally, set a point to fix how far away the dimension is to be from the angle. The angle is measured and the dimension is added to your drawing.

Hint: The options available in the Dimension Angle command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Example: Measure the angle of a three-point line.

Select the DIMENSION ANGLE command. Set a point at the corner of your three-point line for the center of the angle. Next, set a point on the "lower" branch of the line. This is important because the DIMENSION ANGLE command measures positive angles. If you start with the top branch and move to the lower, then DesignCAD measures the obtuse angle around the outside of the three-point line.

Set a third point on the "upper" branch of the line and move the cursor away from the line. Notice the rubber-band dimension line and text box drawn on the screen. When the dimension is where you want it, set the final point.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

Dimension Arc Command

Menu:	DIMENSION
Menu Command:	DIMENSION ARC
Toolbox Icon:	
Point 1:	Arc to be dimensioned
Point 2:	Text location

The Dimension Arc command dimensions the linear length of an arc from endpoint to endpoint.

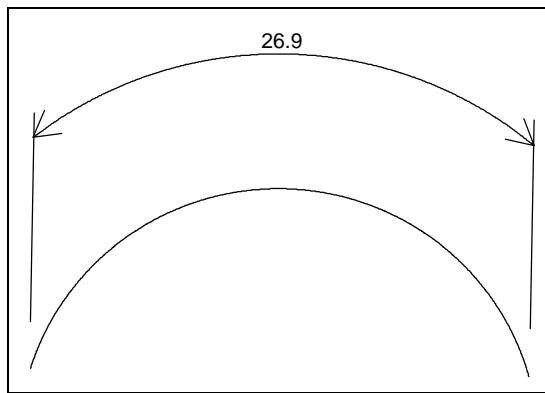
Using the Command

Choose the **DIMENSION ARC** command in the Toolbox. To have the dimension drawn with the same options as another dimension in the drawing, click the **SAME AS** button. Set a point on the dimension in the drawing. Set a point on the arc to be measured. After the point is set, a rubber-band line shows how the dimension will be drawn. Set a point for the location of the text information. The linear length of the arc will be calculated and inserted into the drawing at the last point.

Hint: The other options available in the Dimension Arc command are also available in the Dimension command, and are accessed the same way. For details, refer to the “Dimension Command.”

Example: Draw a dimension of an arc.

First, select **DIMENSION ARC** from the **DIMENSION** menu. Then set a point on the arc to be dimensioned. Set a second point for the text location. The dimension of the arc will be calculated and inserted into the drawing.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the **OPTIONS** command from the **OPTIONS** menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: **Dimension Command, Dimension Angle Command**

Dimension Baseline Command

Menu:	DIMENSION
Menu Command:	DIMENSION BASELINE
Toolbox Icon:	
Point 1-n:	Points to be measured
Point n+1:	Location for dimension information

The Dimension Baseline command draws dimensions using several points along a baseline. Points are set for each measurement position. The distance measured is the vertical or horizontal distance to the measurement points. Each distance is measured from the first point.

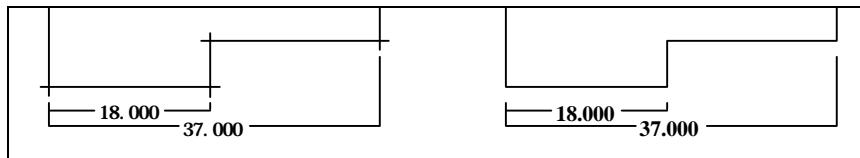
Using the Command

Choose the **DIMENSION BASELINE** command from the Toolbox. Select the **HOR** or **VER** option in the dialog box to determine whether the horizontal or vertical distance will be measured. Next set a point for the base measurement point. Set points for the other measurement points. A rubber-band line shows how the dimension will be drawn. Set a point for the location of the dimension text. Click the middle mouse button or press **Enter**. The dimension is drawn using the points.

Hint: The other options available in the Dimension Baseline command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Example: Dimension the baseline of a box.

Select the **DIMENSION BASELINE** command from the Main Toolbox. Choose the **HOR** option in the dialog box. Set a point for the baseline on the left endpoint of the bottom line of the box. Set two more points for the distance of the bottom line of the box. Set a point in the center and below the line for the text. Press **Enter**. The dimension is drawn using the points.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the **OPTIONS** command from the **OPTIONS** menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: *Dimension Command*

Dimension Chamfer Command

Menu:	DIMENSION
Menu Command:	DIMENSION CHAMFER
Toolbox Icon:	
Point 1:	Line to be dimensioned
Point 2-n:	Extension line
Point n+1:	Text location

The Dimension Chamfer command draws the dimension for a chamfered line. This calculated distance is the vertical or horizontal distance between the two corners of the chamfer.

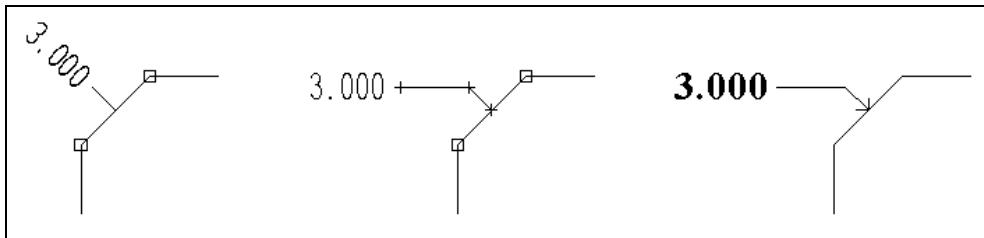
Using the Command

Choose the DIMENSION CHAMFER command in the Main Toolbox. A rubber-band line shows how the dimension will be drawn. Set one or more points for the extension line. Set a point for the text location. Click the middle mouse button or press **Enter** to end the command. The dimension information for the chamfer is drawn at the last point.

Hint: The options available in the Dimension Chamfer command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Example: Dimension the chamfer depth of a line.

Chamfer a line with the CHAMFER command. Then select the DIMENSION CHAMFER command from the Main Toolbox. Set a point on the chamfer line. Set a point for a corner in the extension line. Set another point for the location of the text. Press **Enter**. The dimension information is drawn at the last point.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

Dimension Coordinate Command

Menu:	DIMENSION
Menu Command:	DIMENSION COORDINATE
Shortcut Key:	F11
Toolbox Icon:	
Point 1:	Reference point
Point 2:	Point to be dimensioned
Point 3:	Location for dimension information

The Dimension Coordinate command draws coordinate dimensions relative to a base point. The first point is the origin, or reference point, from which subsequent points are dimensioned. After the base point is set, pairs of points are set. The first point of each pair is the measurement point; the second is the text location.

If the second point in the pair of points is set above or below the first point, the X-value, or horizontal distance, from the reference point will be placed in the drawing. If the second point is set to the right or left of the first point, the Y-value, or vertical distance, from the reference point will be placed into the drawing. Finally, if the second point in a pair of points is set closer or further away along the Z-axis, the relative Z coordinate is placed into the drawing.

Using the Command

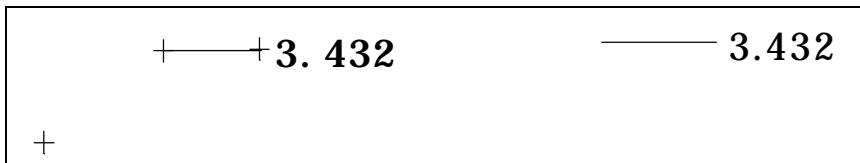
Choose the DIMENSION COORDINATE command in the Main Toolbox. Set a point for the origin, or reference point. Set a point at the coordinate to be measured. Set a point for the text location. Click the mouse or press **Enter** to end the command.

Hint: The options available in the Dimension Coordinate command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

The dimension is measured horizontally, vertically, or along the Z axis from the first point to the second point depending on the relative location of the third point to the second. The dimension text is drawn at the last point.

Example: Dimension a Y coordinate relative to the Origin of the drawing.

Choose the DIMENSION COORDINATE command in the Main Toolbox. Use the POINT XYZ command to set a point (Point 1) for the reference point for the dimension at **0,0,0** (the drawing's origin). Move the cursor along the X and Y axes and set Point 2 at the coordinate to be measured. Set another point for the text location to the right of Point 2. Click the middle mouse button or press **Enter** to end the command.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

Dimension Diameter Command

Menu: DIMENSION

Menu Command: DIMENSION DIAMETER



Toolbox Icon:

Point 1: Circle or arc to be dimensioned

Point 2: Location for dimension information

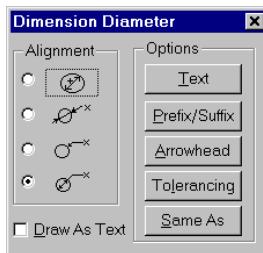
The Dimension Diameter command draws a diameter dimension for a circle or arc. The text and arrowhead can be drawn outside the circle, inside the circle, or with an extension line.

Using the Command

Choose the DIMENSION DIAMETER command in the Toolbox. Set a point on the circle or arc to be dimensioned. After the first point is set, a rubber-band line shows how the dimension will be drawn. Set a point for the location of the text information. The diameter of the circle is displayed at the last point.

Options

The Dimension Diameter command has four options in the dialog box for the extension lines and arrowheads.



Text Inside Circle

- This option (above, far left) draws the dimension text and arrowheads inside the circle or arc.

Pre-Defined Extension Line

- This option (above, center left) draws the dimension text and arrowheads outside the arc or circle.

Custom Extension Line

- This option (above, center right) draws the dimension text and arrowheads outside the arc or circle with a custom extension line.

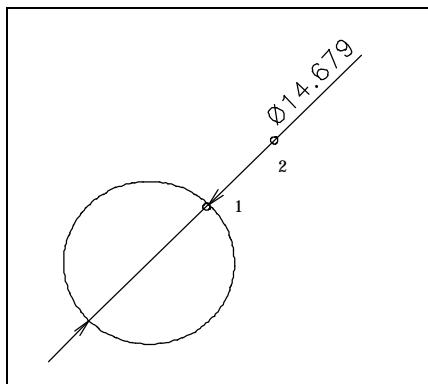
Dimension Inside Circle

- This option (above, right) draws the arrowheads inside the arc or circle.

Hint: The other options available in the Dimension Diameter command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Example: Determine the diameter dimension of a circle.

Select the DIMENSION DIAMETER command from the Main Toolbox. Set a point on the circle. Set a second point for the text to the right of the circle. The diameter dimension is inserted into the drawing.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

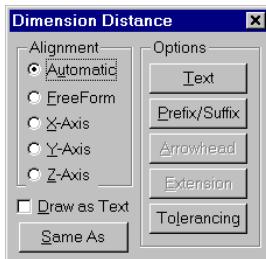
Dimension Distance Only Command

Menu:	DIMENSION
Menu Command:	DIMENSION DISTANCE ONLY
Toolbox Icon:	
Point 1:	Beginning of distance to be measured
Point 2:	End of distance to be measured
Point 3:	Location of dimension text

The Dimension Distance Only command measures a distance horizontally, vertically, or at any angle, but does not add extension lines or arrowheads.

Using the Command

Choose the DIMENSION DISTANCE ONLY command in the Main Toolbox or the DIMENSION menu. You may choose which direction you wish to measure by selecting AUTO, FREE, X, Y or Z.



Hint: The other options available in the Dimension Distance Only command are also available in the Dimension command, and are accessed the same way. For details, refer to the “Dimension Command.”

Set a point for the beginning of the dimension. After the first point is set, a rubber-band box shows where the dimension text will be drawn. Set a point for the end of the dimension. Set a point for the location of the text information. The dimension is drawn without arrows or extension lines.

Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: *Dimension Command*

Dimension Extended Command

Menu: DIMENSION

Menu Command: DIMENSION EXTENDED

Toolbox Icon:



Point 1-n: Points to be measured

Point n+1: Location for dimension information

The Dimension Extended command draws successive dimensions extended along several points. Points are set for each measurement position. The distance measured is the vertical or horizontal distance between the measurement points. Each distance is measured separately between each successive pair of points.

Using the Command

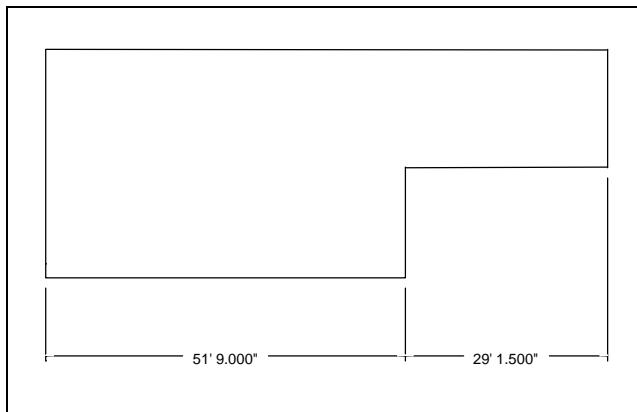
Choose the DIMENSION EXTENDED command from the Main Toolbox. Select the HOR or VER option to determine whether the horizontal or vertical distance will be measured.

Hint: The other options available in the Dimension Extended command are also available in the Dimension command, and are accessed the same way. For details, refer to the “Dimension Command.”

Next, set a point for each measurement position. Then set a point for the location of the dimension information. Press **Enter** to end the command. Dimensions will be drawn for the measurement points.

Example: Dimension a wall.

Select the **DIMENSION EXTENDED** command from the **DIMENSION** menu or the Main Toolbox. Choose the **x** box option in the dialog box. Set a point on the bottom-left corner of the wall. Set a second point near the middle of the wall, along the bottom. Set a third point on the bottom-right corner of the wall. Set a fourth point for the text below the wall. Press **Enter**. The dimension is added to the drawing in two sections.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the **OPTIONS** command from the **OPTIONS** menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

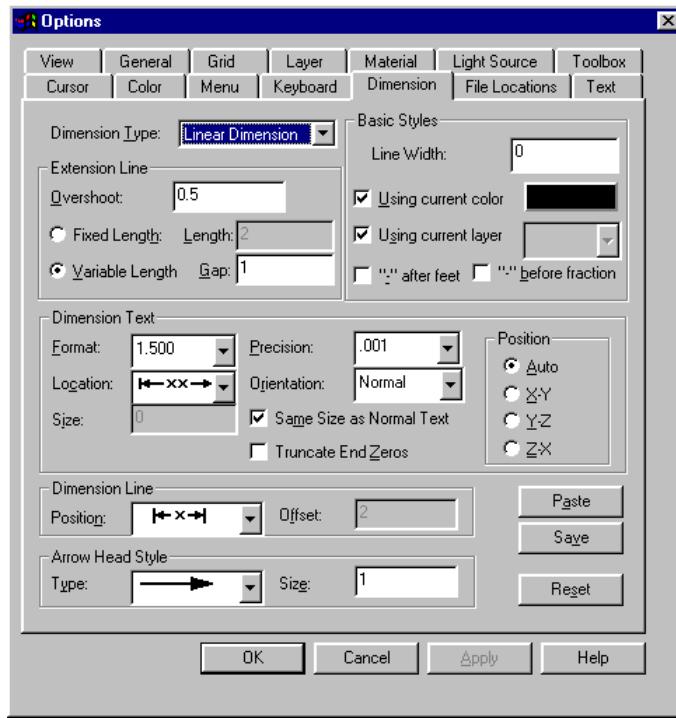
Dimension Options

Menu:	OPTIONS
Menu Command:	OPTIONS

The Dimension Options folder gives you control over the many options in DesignCAD's 12 dimension commands, plus the Pullout and Balloon commands, all from one menu.

Using the Command

Choose the **OPTIONS** command from the **OPTIONS** menu, and then click the **DIMENSION** tab to bring up the Dimension Options folder. Click in the Current Dimension Type box and select the type of dimension command for which you want to set the options.



Once the settings are changed, click **OK** to accept them.

Dimension Line Width

This option determines the width of the lines used to draw dimensions.

Using Current Color

When this option is selected, dimension text and extension lines are drawn with the current drawing color. If you are drawing with several different colors and want all dimension text to be drawn with the same color, uncheck the **USING CURRENT COLOR** checkbox and click the **COLOR** box; the Color Palette appears. Click on the color you want to use for dimension text and click **OK**.

Using Current Layer

This option determines the layer in which the dimension text and extension lines are drawn. When the **Using Current Layer** option is checked, all dimension text is drawn in the active layer. To draw all of the dimension text in the same layer of a multi-layer drawing, uncheck the **USING CURRENT LAYER** box and click the **LAYER** box. Select the layer you want dimension text to be drawn in and click **OK**.

Paste

After setting the options for a particular dimension command, click on the **PASTE** button to apply the same options to similar dimensioning commands. If you have set the options for a dimensioning command that measures an angular dimension, the same options will be used for all dimensioning commands that make angular measurements. Likewise, if you have set the

options for a dimensioning command that measures a linear dimension, the same options will be used for all dimensioning commands that make linear measurements.

Save

If you want to save the changes to the next drawing session, click the **SAVE** button before you click **OK**.

Reset

The **RESET** button restores all of the options for the command specified in the Current Dimension Type box to their default values.

For more detailed Dimension Options information, refer to the individual entries listed in the "Command Reference" section of this manual. Dimension entries include:

- Balloon Command
- Dimension Command
- Dimension Angle Command
- Dimension Arc Command
- Dimension Baseline Command
- Dimension Chamfer Command
- Dimension Coordinate Command
- Dimension Diameter Command
- Dimension Distance Only Command
- Dimension Extended Command
- Dimension Progressive Command
- Dimension Radius Command
- Dimension Radius Progressive Command
- Pullout Command

Dimension Progressive Command

Menu:	DIMENSION
Menu Command:	DIMENSION PROGRESSIVE
Toolbox Icon:	
Point 1:	Base point
Points 2-n:	Points to be measured
Point n+1:	Location for text

The Dimension Progressive command draws the dimensions of a line progressively from a base point. The distance measured is the vertical or horizontal distance between each measurement point and the base point.

Using the Command

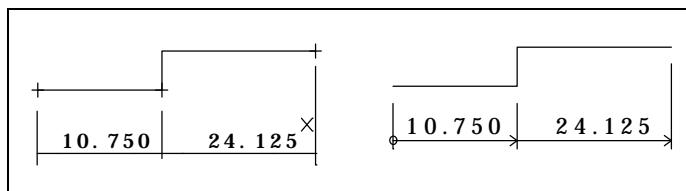
Choose the DIMENSION PROGRESSIVE command from the Main Toolbox or the DIMENSION menu. Select HOR or VER to determine whether the horizontal or vertical distance will be dimensioned.

Hint: The other options available in the Dimension Progressive command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Set a point for the base measurement point. Set points for the other measurement points. A rubber-band line shows how the dimension will be drawn. Finally, set a point for the location of the dimension text. Then press **Enter** to end the command. The dimension is drawn from the base point to each measurement point.

Example: Draw progressive dimensions for an object.

Select the DIMENSION PROGRESSIVE command. Choose the HOR option to measure the horizontal distance for the object you are going to dimension. Next, set a base point at one end of the object. Set additional points as needed for the steps you want to measure. Then set a point for the text and press **Enter**. The progressive dimensions of the object are calculated from the first point and inserted into the drawing.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: *Dimension Command*

Dimension Radius Command

Menu:	DIMENSION
Menu Command:	DIMENSION RADIUS
Toolbox Icon:	
Point 1:	Circle or arc to be dimensioned
Point 2:	Location for dimension information

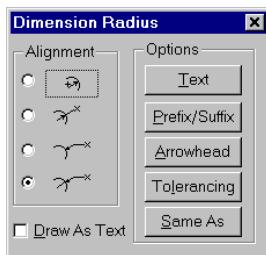
The Dimension Radius command dimensions the radius of a circle or arc.

Using the Command

Choose the **DIMENSION RADIUS** command from the Main Toolbox or the **DIMENSION** menu. To have the dimension arrowhead drawn inside the circle, click the **PRE-DEFINED EXTENSION** button in the dialog box.

Options

- The Dimension Radius command has four options in the dialog box for the extension lines and arrowheads.



Dimension Inside Circle

- This option (above far left) draws the dimension text and arrowheads inside the circle or arc.

Pre-Defined Extension Line

- This option (above center left) draws the dimension text outside the arc or circle, with the arrow inside.

Custom Extension Line

- This option (above center right) draws the dimension text and arrowheads outside the arc or circle with a custom extension line.

Dimension Outside Circle

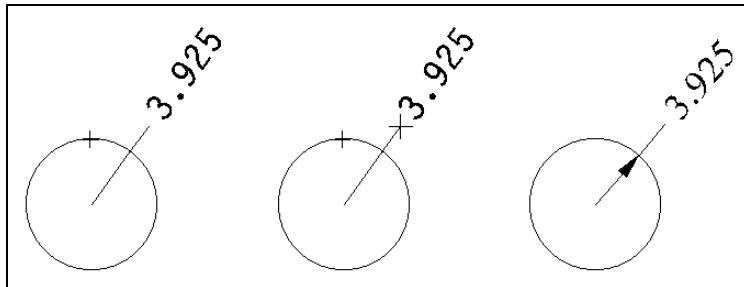
- This option (above right) draws a reference line from the center of the arc or circle to the perimeter, and the text and arrowheads are placed outside the arc or circle.

Hint: The other options available in the Dimension Radius command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Set a point on the circle or arc to be dimensioned. After the first point is set, a rubber-band line shows how the dimension will be drawn. Set a point for the location of the text information. The radius of the circle is displayed at the last point.

Example: Insert a dimension with the Dimension Radius command.

Choose the **DIMENSION RADIUS** command. Set a point on the circle or arc to be dimensioned. Set a second point for the location of the text. The dimension information is calculated and inserted into the drawing.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: Dimension Command

Dimension Radius Progressive Command

Menu: DIMENSION
 Menu Command: DIMENSION RADIUS PROGRESSIVE
 Toolbox Icon: 
 Point 1: Base point
 Points 2-n: Points to be measured
 Point n+1: Location for text

The Dimension Radius Progressive command draws radius dimensions progressively. Points are set for each measurement position. The dimension of the radius is calculated and inserted into the drawing using progressive measurements. Each distance is measured from the base point.

Using the Command

Choose the DIMENSION RADIUS PROGRESSIVE command from the Main Toolbox or the DIMENSION menu.

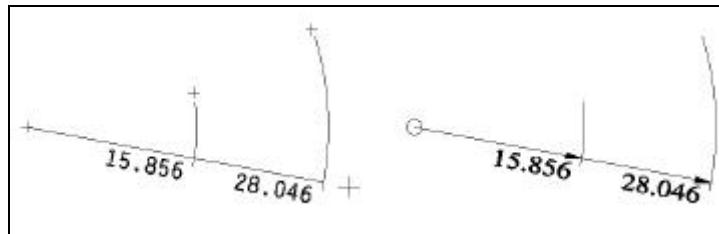
Hint: The other options available in the Dimension Radius Progressive command are also available in the Dimension command, and are accessed the same way. For details, refer to the "Dimension Command."

Set a point for the base point. Set points for the other measurement points. Set a point for the text location. Click the middle mouse button or press **Enter**.

Example: Draw the progressive dimensions of a radius.

First, select the DIMENSION RADIUS PROGRESSIVE command from the Main Toolbox or the DIMENSION menu. Next, set a base point at the center of the object to be dimensioned. Set one or more points for measurement. Then set a point for the text location. Press **Enter**, and the dimension

of the radius is calculated from the first point to each successive point and inserted into the drawing.



Hint: The Dimension Options folder gives you control over all options in DesignCAD's 12 dimension commands from a single menu. To use the Dimension Options folder, choose the OPTIONS command from the OPTIONS menu, and then click the Dimension tab. Select the type of dimension command for which you would like to set the options from the Current Dimension Type box.

See Also: *Dimension Command*

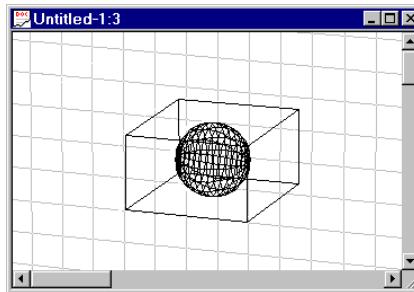
Display Grid Command

Menu:	OPTIONS
Menu Command:	DISPLAY GRID

The Display Grid command shows a grid on the drawing screen which can be used as a reference for drawing new objects or comparing the size of existing objects.

Using the Command

Choose DISPLAY GRID in the OPTIONS menu. The Grid immediately becomes visible on the drawing screen. You can change the size of the grid units, the extent of the grid, the grid plane, the grid type, and the grid color in the GRID options folder available through the OPTIONS command in the OPTIONS menu.



See Also: *Grid Options*

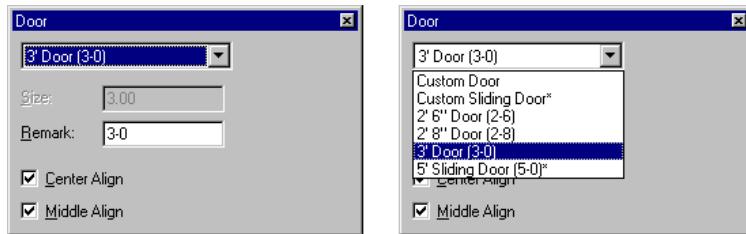
Door Command

Menu: DRAW
 Menu Command: DOOR
 Point 1: location of the center of the door or one end of the door
 Point 2: orientation of the door

The Door command inserts a standard architectural symbol for a door.

Using the Command

Choose the DOOR command from the DRAW menu. The Door dialog box appears.



Select the desired door size and type from the list box. If you have chosen the Custom Door or Custom Sliding Door option, specify the size in the SIZE box. By default, the door size appears in the REMARK box unless Custom is selected. However, any information can be entered in the Remark box to be inserted into the drawing with the door. Check the CENTER ALIGN box to align the door by the center or uncheck it to align the door by one end. Check the MIDDLE ALIGN box to align the middle of a double line.

Set the first point for the door. A rubber-band door appears, showing how the door would look with the second point set at the current cursor location. When the door is positioned to your liking, set the second point. The Line or Double Line into which the door is placed will automatically be cut and capped.

Double Line Mode Command

Menu: OPTIONS
 Menu Command: DOUBLE LINE MODE

The Double Line Mode command acts as a toggle to turn Double Line Mode on or off. When simple 2-D entities are drawn in Double Line Mode, DesignCAD draws two parallel entities at the same time. Double Line Mode simplifies several common types of drawings, including house plans and land plots.

Using the Command

Choose the DOUBLE LINE MODE command from the OPTIONS menu. DesignCAD enters Double Line Mode. Simple 2-D entities such as arcs, circles, curves, and lines will automatically be drawn with duplicate parallel entities.

Use the Double Line bar to control where and how the parallel entities are drawn.

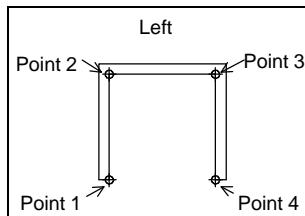
Double Line Width: <input type="text" value="0.5"/>	Align: <input type="button" value="Center"/>	Cap: <input type="button" value="Both"/>	<input type="checkbox"/> Filled	<input type="checkbox"/> AutoTrim	A	A
-----------------------------------------------------	----------------------------------------------	------------------------------------------	---------------------------------	-----------------------------------	----------	----------

Double Line Width

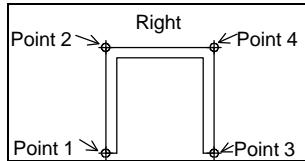
Sets the width of the entity (the amount of space between the two parallel entities) in drawing units.

Align

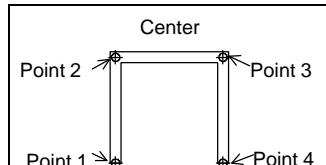
Select Center, Left, or Right to set the position of the width of the double entity. If Left is selected, the width of the entity will extend to what would be your left if you were standing on the first point set for the entity facing the second point set for the entity.



If Right is selected, the width of the entity will extend to what would be your right if you were standing on the first point set for the entity, facing the second point of the entity.



If Center is selected, the width of the entity will extend an equal distance on both sides of the points.



Note: If the double entity is drawn with the Left or Right Align option selected, you will be able to snap to both sides of the finished double entity. If the double entity is drawn with the Center option selected, you will be able to snap to both sides and the center of the double entity.

Cap

Select Both, Begin, End, or None to specify which ends of the double entity are capped or left open.

Filled

Check the Filled option to draw a filled entity, or leave it unchecked to draw a hollow entity.

Auto Trim

If the Auto Trim option is checked, the double entity will automatically be trimmed where it overlaps itself or another double entity.

Note: The Explode command can be used to explode a double entity into vectors. If you are going to cross one double entity over another, explode the existing double entity and uncheck the Auto Trim box before drawing the second double entity to ensure that all four entities will be visible in the intersection.

Apply to Selected Double Lines

Click **APPLY TO SELECTED DOUBLE LINES** to apply the current options to selected double entities in the drawing.

Apply to All Double Lines

Click **APPLY TO ALL DOUBLE LINES** to apply the current options to all double entities in the drawing.

Same As

Click the **SAME AS** button to match the parameters of an existing double entity. Just click on the **SAME AS** button, then click on the double entity already in the drawing that has the properties you want to assign to the options in the Double Line bar.

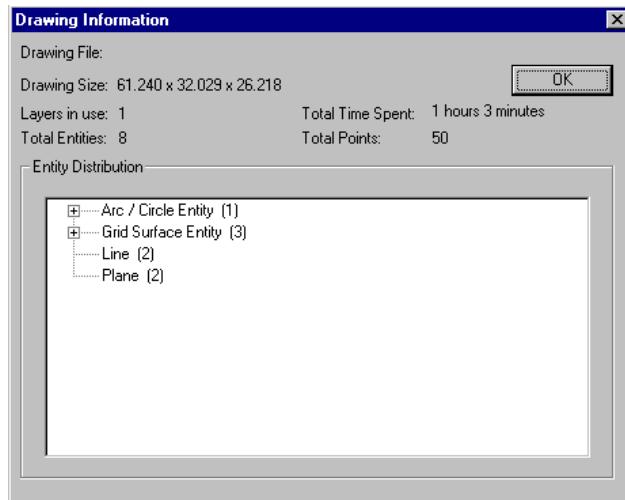
Set the points for the double entity. You will see a rubber-band double entity connected to each point you set. When you have set the points, press **Enter** or double-click the mouse to end the command.

See Also: *Convert to Double Lines Command*

Drawing Info Command

Menu:	HELP
Menu Command:	DRAWING INFO

The Drawing Info command displays the number of entities and points in a drawing, along with various other information.



Duplicate Command

Menu: EDIT
 Submenu: SELECTION EDIT
 Menu Command: DUPLICATE
 Shortcut Key: **N**
 Point 1: Handle 1 sets location for the copy
 Point 2: Handle 2 sets angle and scale for the copy (optional)
 Point 3: Handle 3 sets orientation for the copy (optional)

The Duplicate command makes a copy of the current selection. The command does not use the Windows Clipboard, so any contents in the Clipboard are not erased or changed.

Using the Command

Select an object or objects, and then choose the DUPLICATE command. One, two, or three points may be used to specify the location where the copy is to be placed.

The first handle of the selection is positioned at the first point set with this command. If you have set two or three drawing handles, then you can place and scale the duplicate by those points. Otherwise, the first point you place will make a duplicate with the same orientation and size as the selection.

If a second point is used, the selection's size and angle are adjusted so that the secondary handle is located at the second point. If only one point is used, the selection will be placed at its original size and angle.

If a third point is used, the selection will be positioned at an angle so that the three Block handles lie on the same plane as the three points set.

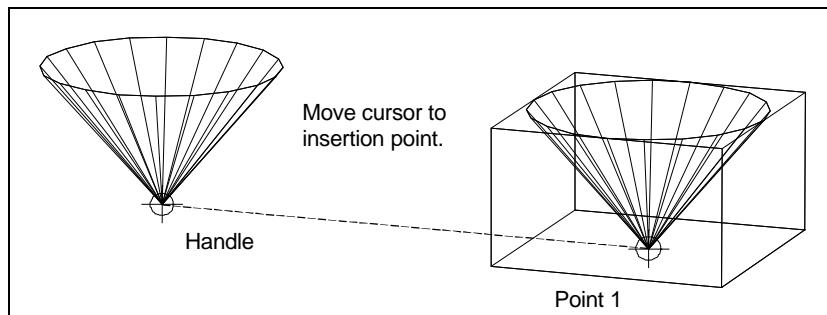
Note: In order to use the second selection handle to change the size of the selection, you must uncheck the Use Original Scale checkbox.

You can also activate this command by moving the cursor onto the primary handle of an object, then holding down **Ctrl** and clicking the left mouse button to lock the Duplicate command.

This command is like the Move command, except that the selection is copied instead of moved.

Example: Make a copy of an object in your drawing.

Select the object and choose the DUPLICATE command. Move the cursor to the location for the copy. Set a point for the insertion. If you have set more than one handle for the object, then set a point for each handle.



Ellipse Command

Menu: DRAW

Submenu: CIRCLES

Menu Command: ELLIPSE

Toolbox Icon:



Point 1: Center of the ellipse

Point 2: One axis of the ellipse

Point 3: Point through which the ellipse passes

The Ellipse command draws an ellipse or oval. Points are set for the center and axes.

Using the Command

Choose the ELLIPSE command in the Main Toolbox. Set a point for the center of the ellipse. After the first point is set, a rubber-band ellipse shows how the ellipse will be drawn. Set a point for one axis of the ellipse. Set a point through which the ellipse passes. An ellipse will be drawn passing through the second and third points with its center at the first point.

The ellipse can be saved in the drawing as one of three forms:

1. **Circle:** Stored as an actual circle in the drawing.
2. **Plane:** Stored as a circular plane with 36 sides which can be shaded.
3. **Line:** Stored as a line entity.



When the **CIRCLE** option is selected, the circle is saved in the drawing as a Circle entity. This is the way ellipses are normally saved with DesignCAD 3D MAX.

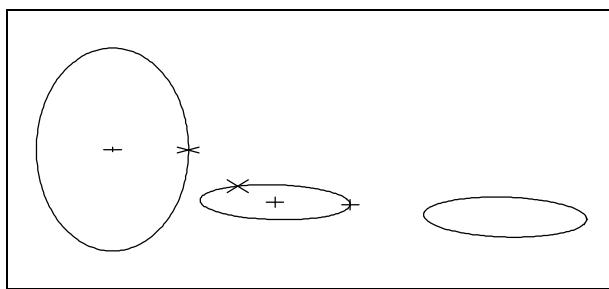
The ellipse can be saved as a plane so it can be shaded, subtracted, etc.

The ellipse can also be saved as a line. This makes it possible to scale the ellipse or treat it as a line entity with other commands.

Usually, you will want to save the ellipse as a Circle entity or as a Plane entity.

Example: Draw an ellipse.

Choose the **ELLIPSE** command. Set a point for the center of the ellipse. Then set a point up and to the left of the first point. This point is one axis. Next, set a third point to the right of the second point. The ellipse is drawn, passing through this point and using all three points.



See Also: Elliptical Arc Command

Elliptical Arc Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	ELLIPTICAL ARC
Toolbox Icon:	
Point 1:	Center of the ellipse
Point 2:	Beginning of the arc
Point 3:	End of the arc

The Elliptical Arc command draws an elliptical arc, or part of an ellipse or oval, between two points.

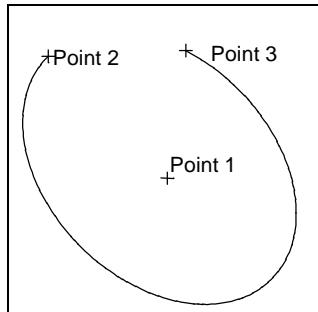
Using the Command

Choose the **ELLIPTICAL ARC** command in the Main Toolbox. Set a point for the center of the arc. Set a point for the beginning of the arc. After the second point is set, a rubber-band arc shows

how the arc will be drawn. Set a point for the end of the arc. An elliptical arc is drawn from the second point, counterclockwise around the ellipse to the third point.

Example: Draw an elliptical arc.

Choose the ELLIPTICAL ARC command. Set a point for the center of the arc. Next, set a point to the left of the first point. Set a third point to the right of the first point. An elliptical arc is drawn through these points.



See Also: Ellipse Command

Elliptical Arc (Center, Axes, Angles) Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	ELLIPTICAL ARC (CENTER, AXES, ANGLES)
Point 1:	Center of the elliptical arc
Point 2:	Major axis
Point 3:	Minor axis
Point 4:	Beginning of the arc
Point 5:	End of the arc

The Elliptical Arc (Center, Axes, Angles) command draws an elliptical arc, or part of an ellipse or oval, using a point for the major axis, a point for the minor axis, a point for the beginning of the arc, and a point for the end of the arc.

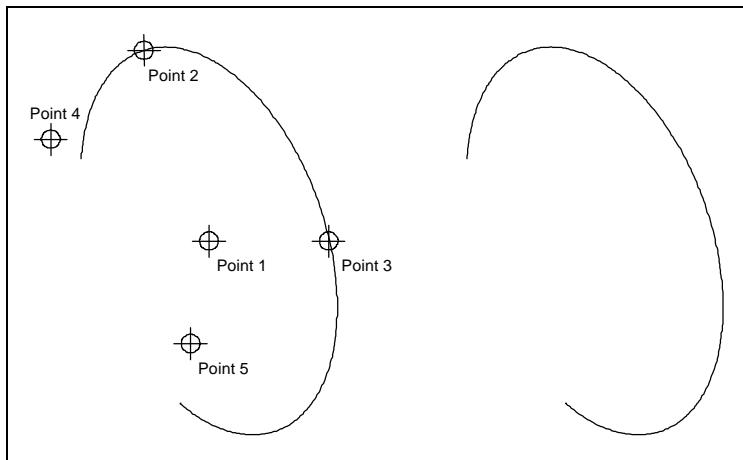
Using the Command

Choose the ELLIPTICAL ARC (CENTER, AXES, ANGLES) command from the ARCS submenu of the DRAW menu. Set a point for the center of the arc. Set a point for the major axis and a point for the minor axis. Set a point for the beginning of the arc. After the fourth point is set, a rubber-band arc shows how the arc will be drawn. Set a point for the end of the arc. The elliptical arc is drawn.

Example: Draw an elliptical arc.

Choose the ELLIPTICAL ARC (CENTER, AXES, ANGLES) command. Set a point for the center of the arc. Next, set a point above and slightly to the left of the first point for the major axis of the elliptical arc. Set a third point to the right of the first point for the minor axis of the elliptical arc. Set a fourth point to the left and slightly above the first point for the beginning span angle. Finally,

set a fifth point below and slightly to the left of the first point for the ending span angle. An elliptical arc is drawn using these points.



See Also: *Ellipse Command, Elliptical Arc Command*

Erase Command

Menu:	EDIT
Menu Command:	ERASE
Shortcut Key:	Del

The Erase command deletes selected entities from the drawing. You must first select the entity you want to erase, and then choose the Erase command.

Example: Remove an object from a drawing.

Select an item in the drawing you want to erase. Then choose the ERASE command. The item is erased from the drawing.

Erase Last Command

Menu:	EDIT
Menu Command:	ERASE LAST
Shortcut Key:	Ctrl+E

The Erase Last command deletes the most recently drawn entity from the drawing.

Using the Command

When you choose the Erase Last command, the most recently drawn entity is removed from the drawing. Each subsequent time you use this command, it erases the next most recently drawn entity, working its way from last to first entity drawn in that session.

The Erase Last command pays attention only to the order in which entities were drawn originally. It does not acknowledge modifications to existing entities or undo them, but erases entire entities.

Example: Erase the last two items added to the drawing.

Choose the ERASE LAST command twice. DesignCAD erases the last object and the next to the last object.

Hint: To undo a modification, use the UNDO command.

Exclusive Or Command

Menu: SOLIDS
Menu Command: EXCLUSIVE OR
Point 1: First Solid
Point 2: Second Solid

The Exclusive Or command is used to "subtract" the volume in common between two solids. If a portion or portions of both of the solids remain, the portions are still recognized as a part of their original solid.

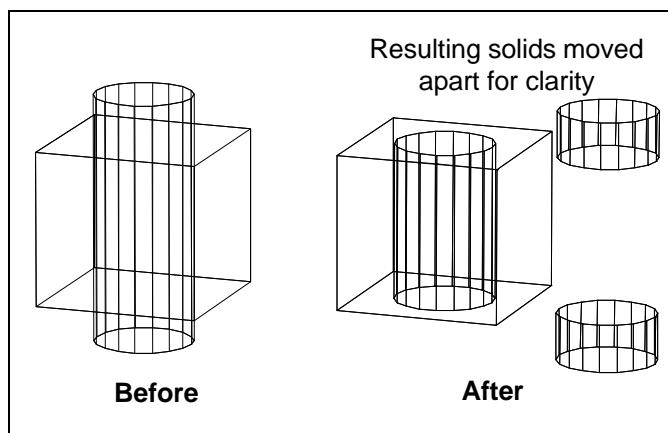
Using the Command

Choose the Exclusive Or command. Set a point on each of the solids. The common volume between the two solids is removed. The only portions that remain are portions that were exclusive to one or the other of the two original solids.

Note: For best results, the solids should overlap rather than meet exactly at a face.

Example: Subtract the common volume from an intersecting box and cylinder.

Select the EXCLUSIVE OR command and set a point on each of the solids. DesignCAD removes the shared volume and redraws the remaining portions.



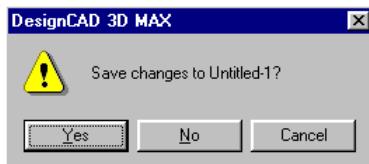
Exit Command

Menu:	FILE
Menu Command:	EXIT
Shortcut Key:	Alt+F4

The Exit command closes DesignCAD 3D MAX.

Using the Command

Choose the EXIT command. If you have any unsaved drawings open, the program gives you the opportunity of saving each one before the program closes.



Choosing YES activates the Save As command. Choosing NO closes the program (or prompts you to save the next drawing if more than one were open). Choosing CANCEL returns you to the DesignCAD drawing screen with no change.

Explode Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	EXPLODE

The Explode command converts any selected composite objects into separate entities. This command affects grids, dimensions, and symbols. Afterwards, the parts of the former composite objects can be manipulated individually.

Symbols are broken apart into separate drawing entities, just as if they had been drawn originally right in the current drawing. Grids are converted into planes, and planes are converted to vectors. Dimensions are converted into text, vectors, and arrows.

If a group is selected when the Explode command is chosen, objects in the group will be exploded as if they were not a part of a group, but the group will remain intact. To select individual entities, it will still be necessary to use the Group Explode command to explode the group. To explode a group so individual objects may be selected, but still remain composite objects, use the Group Explode command instead of the Explode command.

Using the Command

Select the object(s) to be exploded. Choose EXPLODE from the EDIT | SELECTION EDIT submenu. The entities can be selected separately.

Example: Explode a symbol.

Add a Dimension to a drawing and select the Dimension. Choose EXPLODE from the EDIT | SELECTION EDIT menu. The Dimension will be separated into the entities that make it up.

See Also: Group Explode Command

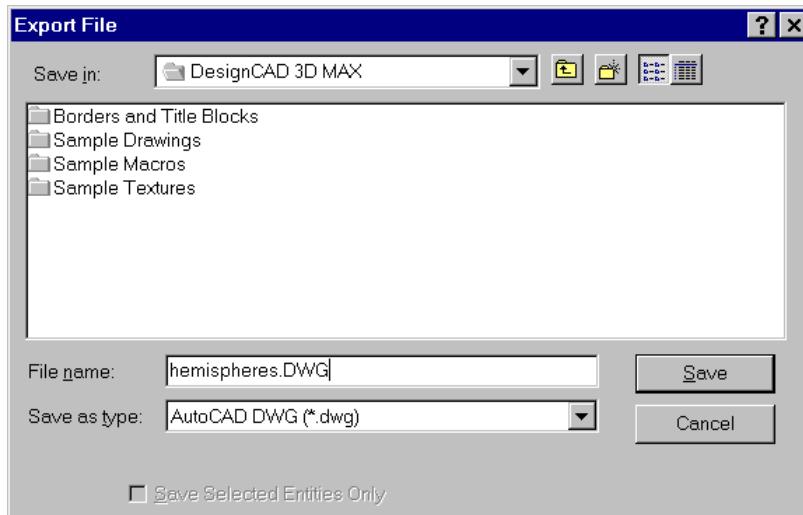
Export Command

Menu:	FILE
Menu Command	EXPORT

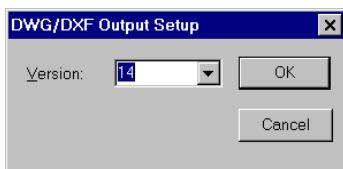
The Export command lets you export files in several formats.

Using the Command

Select EXPORT from the FILE Menu. The EXPORT FILE box appears. In the FILE NAME box enter the name of the file to export. In the SAVE IN box tell the program where to store the exported drawing. In the SAVE AS TYPE box, enter the format of the drawing you want to export. When you have entered the information, choose the SAVE button to save and export the drawing. Choose the CANCEL button to return to the drawing without exporting it.



When exporting DWG or DXF format, the DWG/DXF OUTPUT SETUP dialog box will appear after the EXPORT FILE box closes. Select the desired version from the VERSION: box. Click the OK button and the file will be exported. Click the CANCEL button to return to the drawing without exporting it.



DWG

DesignCAD can export your drawing in DWG format.

AutoCAD Drawing Interchange (DXF)

DXF files can be used with many other Windows applications.

Initial Graphics Exchange Specifications (IGES)

The IGES format is a standard format that many CAD systems support.

Windows Metafile (WMF)

Many Windows applications can read Windows Metafiles.

WordPerfect Graphic (WPG)

DesignCAD can export files in the WordPerfect .WPG format.

RenderMan (RIB)

Export to the RenderMan rendering package.

VRML (WRL)

The WRL format is for Internet graphic files that can be read by any 3D web browser.

Example: Export a drawing as a Windows Metafile.

Choose the EXPORT command. The SAVE box appears. Designate .WMF format in the SAVE AS TYPE box. Choose a path and file name for the file and click OK. The drawing is converted into .WMF format and saved. Now you can open a word processing program, desktop publishing program, or other application that imports graphics and import the Windows Metafile into the application.

Extend Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	EXTEND
Toolbox Icon:	
Point 1:	Endpoint
Point 2:	New Location

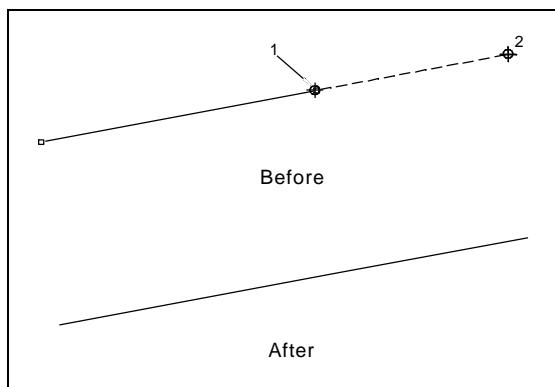
The Extend command extends or shortens a line by moving one of the points defining the end of the line. The line maintains the same angle after it has been extended.

Using the Command

Choose EXTEND from the TRIM/EXTEND submenu in the EDIT menu. Now, set a point on the endpoint to be extended. Move the cursor to the new location for the endpoint. A rubber-band line shows how the extended line will look. Set a point for the new endpoint. The line will be redrawn to use the new endpoint.

Example: Extend a line.

Choose the EXTEND command. Set a point on the endpoint you want to extend. Move your cursor to a new location for the endpoint. A rubber-band box shows how the extended line will look. Then set a point for the new endpoint. The line is redrawn to use the new endpoint.



Extend by Distance Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	EXTEND BY DISTANCE
Point 1:	Endpoint

The Extend by Distance command extends or shortens a line by moving one of the points defining the end of the line a specified linear distance, to a relative X coordinate, to a relative Y coordinate, or to a relative Z coordinate. The line maintains the same angle after it has been extended.

Using the Command

Choose EXTEND BY DISTANCE from the TRIM/EXTEND submenu in the EDIT menu. Enter the distance to be added to the line in the DISTANCE: box. Specify whether the extension for the line is to be measured along the line or with respect to the X-Axis, the Y-Axis, or the Z-Axis by marking the radio button beside the desired option. Now, set a point on the endpoint to be extended. The line will be redrawn to match its new specifications.

Example: Extend a line.

Choose the EXTEND BY DISTANCE command. Enter **15** in the DISTANCE: box. Mark the radio button beside the ALONG LINE option. Set a point on the endpoint you want to extend. The line is extended 15 units.

Extrude Command

Menu: DRAW
 Menu Command: EXTRUDE
 Shortcut Key: X
 Toolbox Icon: 

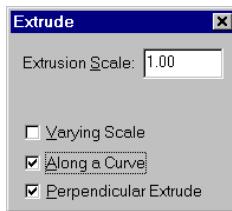
Point 1-n: Distance and direction for the extrusion path

The Extrude command extends a two-dimensional object into three dimensions, connecting it from one point to another with surfaces. For example, you can extrude a floor plan upward to make the walls of a house, or you can extrude a circle to make a pipe or bar.

Using the Command

Select the object to be extruded and choose the EXTRUDE command. Set a reference point and one or more points for the object to be extruded to. The object is extruded so the selection handle is placed at each point set.

You can specify an extrusion scale to make the resulting object increase or decrease in size. The scale is the amount of change in size of the extruded object from the first point to the last.



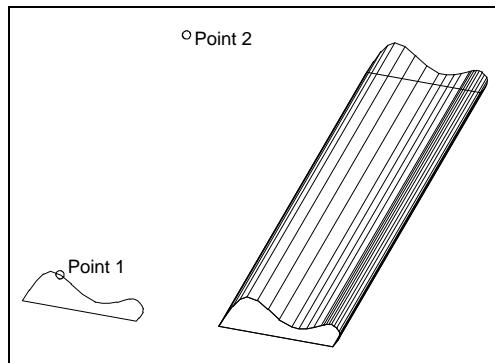
Varying scale can be used to specify the scale at each point along the extrusion, if there is more than one point. If varying scale is used, you can specify the scale for each point. The scale is relative to the extruded object's original size and not necessarily to its size at the previous point.

Example: Select an object used for a mold or cross-section.

Now pick the EXTRUDE command from the DRAW menu. Set two or more points: a reference point and one or more points representing the extrusion path. The path does not have to be in a straight line.

The reference point represents the location of the selected object's handle. The other points represent the distance and direction of the extrusion points. They form a parallel to the actual extrusion path.

The object does not move to the reference point. Instead, the extrusion always starts at the current location of the object. Therefore, it is most convenient just to set the reference point on the handle of the object. Press **Enter** when you have set all the points.



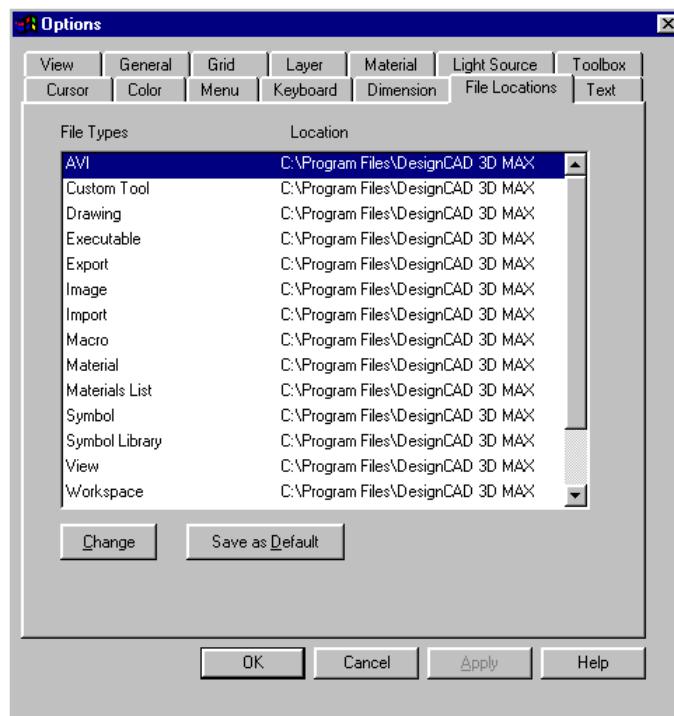
File Location Options

Menu: OPTIONS
 Menu Command: OPTIONS

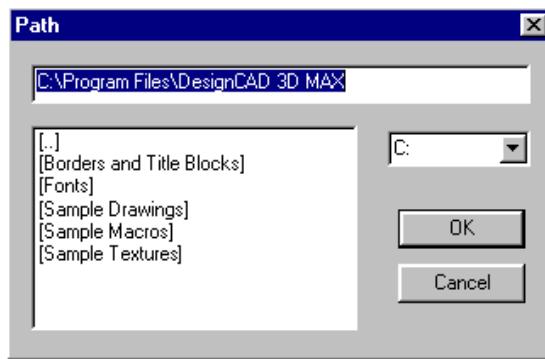
In the File Location Options folder, you can modify where DesignCAD stores the various files it uses and produces. This gives you complete control over file placement, no matter how many hard drives or drive partitions you may have on your computer.

Using the Command

Choose the OPTIONS command from the OPTIONS menu, and then click the FILE LOCATIONS tab to bring up the File Location Options folder.



Scroll through the list of file paths and highlight the file path you want to modify, then click the CHANGE button. The Path box appears.



In the Drive box, click the arrow button to scroll through the list and select the drive that contains the folder or directory you want to use. In the Folder box, scroll through the list and select the name of the folder you want to use. The full path you have selected appears in the box at the top of the Path Selection box. When the path is correct, click OK to change it.

The File Locations Options tab now displays the new path you selected in the list of file paths. To use the new path, click OK.

Now, when you tell DesignCAD to use or save a particular type of file, it will look for or save that file type in the new folder location you selected.

Fillet Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: FILLET
 Shortcut Key: F

Toolbox Icon:

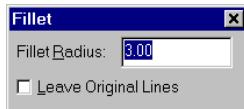
Point 1: A point on one of the lines to be filleted

Point 2: A point on the second line to be filleted

The Fillet command rounds a corner of a plane or two lines.

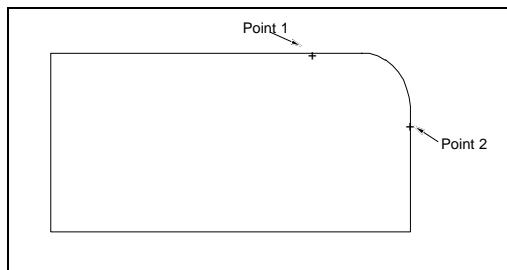
Using the Command

Choose the FILLET command. In the Fillet Radius box in the dialog box enter the radius of the curve to be drawn. Set a point on one of the lines to be filleted, then a point on the other. If the lines are not connected, DesignCAD connects them at the points, using the radius you have set.



Example: Round off the corner of a rectangular plane to a radius of three Drawing Units.

Choose the FILLET command and enter 3 in the FILLET RADIUS box. Set a point near one corner of the plane, and a second point on the other side of the corner. The corner is rounded off to a radius of three.

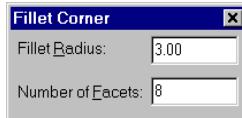


See Also: *Fillet Corner Command*

Fillet Corner Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	FILLET CORNER
Toolbox Icon:	
Point 1:	Corner to be filleted

The Fillet Corner command allows you to round off rectangular corners of solids. You can specify the fillet radius and the number of facets to use on each rounded edge:



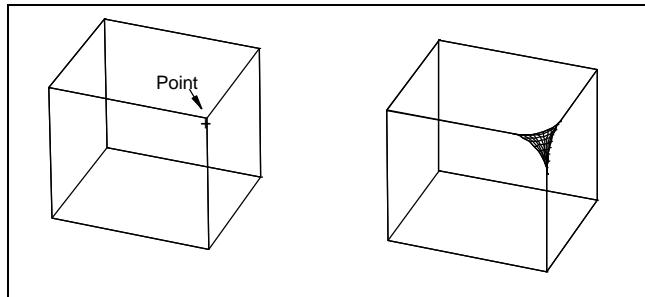
Using the Command

Choose the FILLET CORNER command. In the dialog box enter the amount of the radius in the FILLET RADIUS box and the number of facets in the NUMBER OF FACETS box. Then set a point on the corner to be filleted.

Note: You cannot use this command to round a corner where more than three faces meet, such as the point of a cone.

Example: Round the corner off of a box.

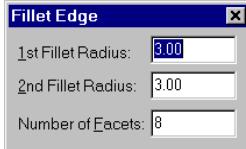
Select the FILLET CORNER command. Move the cursor near the corner you want to fillet and set a GRAVITY point by clicking the right mouse button or pressing the . (period) key. The corner is filleted to the radius specified in the dialog box.



Fillet Edge Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: FILLET EDGE
 Toolbox Icon: 
 Point 1: Edge to be filleted

The Fillet Edge command quickly and easily rounds off rectangular edges of solid objects. You may specify the number of facets for the rounded edge and the radius for each end of the edge to be filleted.



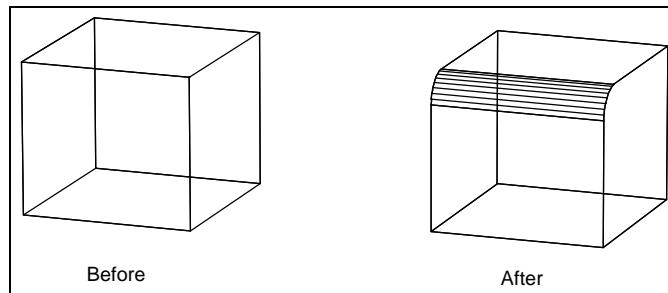
Using the Command

Choose the FILLET EDGE command. The dialog box shows three boxes. Enter the radius for the beginning of the edge in the 1ST FILLET RADIUS box. Enter the radius for the end of the edge in the 2ND FILLET RADIUS box. In NUMBER OF FACETS box enter the number of facets you want the edge to have. Return to the drawing screen. Then set a point on the edge (not at a corner) you want to fillet.

Note: The Fillet Edge command requires flat edges. Once you have filleted an edge of a cube, for example, the adjacent edges no longer meet at a flat face but at a curved surface. So the edges adjacent to a filleted edge cannot also be filleted.

Example: Round off the rectangular edge of a solid.

Choose the FILLET EDGE command and enter the values for the radius of each fillet and the facets in the dialog box. Set a point on the edge to be filleted. DesignCAD fillets the edge according to the values.



Fit to Window Command

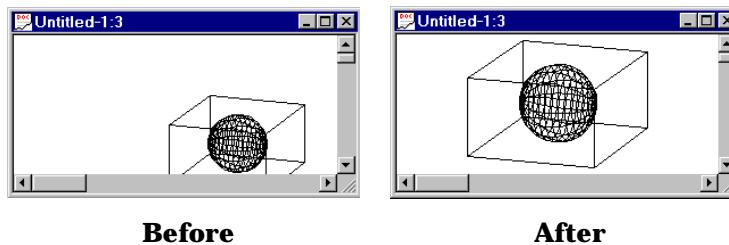
Menu:	VIEW
Menu Command:	FIT TO WINDOW
Shortcut Key:	Ctrl+W
Toolbox Icon:	

This command zooms the active view window so that the entire drawing is centered on the screen with all objects visible.

Using the Command

Choose the FIT TO WINDOW command. The drawing is then centered on the screen with all the objects in view.

Suppose you have several entities in your drawing that are outside of the current view, or a single entity drawing that you want to center and zoom in on. The Fit to Window command brings them all into view, showing you the closest zoom possible of the entire drawing in the current view.



Fit to All Windows Command

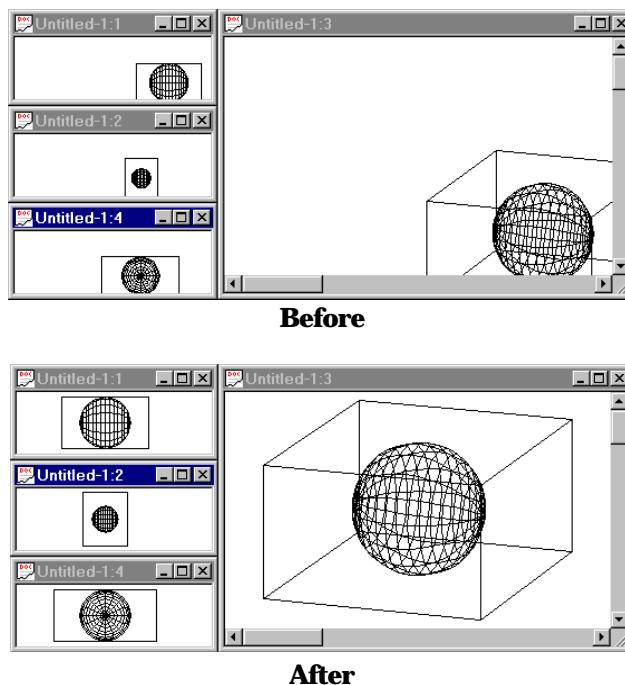
Menu:	VIEW
Menu Command:	FIT TO ALL WINDOWS
Shortcut Key:	Ctrl+Shift+W
Toolbox Icon:	

This command zooms all the view windows so that the entire drawing is centered on the screen with all drawn objects visible.

Using the Command

Choose the FIT TO ALL WINDOWS command. The program then redraws all the objects in the drawing so they are visible in each open window.

For example, if you have several entities in your drawing that are outside of the current view, or a single entity drawing that you want to center and zoom in on, you might want to use the Fit to All Windows command. The Fit to All Windows command brings them all into view, showing you the closest zoom possible of the entire drawing in all active views.



See Also: *Fit to Window Command, Zoom In Command, Zoom Out Command*

Four-Point Copy Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	FOUR-POINT COPY
Point 1:	location for lower-left corner of copy
Point 2:	location for lower-right corner of copy
Point 3:	location for upper-left corner of copy
Point 4:	location for upper-right corner of copy

The Four-Point Copy command copies a selected 2-D entity or group of entities to another location. Four points are used to specify the new location.

Using the Command

To use the command, select the 2-D entity or group of entities to be copied. Choose the FOUR-POINT COPY command. Set four points for the four corners of the new copy of the selection. When the fourth point is set, the new copy of the selection is inserted in the drawing.

See Also: *Four-Point Move Command, Copy Command*

Four-Point Move Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	FOUR-POINT MOVE
Point 1:	new location for lower-left corner of selection
Point 2:	new location for lower-right corner of selection
Point 3:	new location for upper-left corner of selection
Point 4:	new location for upper-right corner of selection

The Four-Point Move command moves a selected 2-D entity or group of entities to another location. Four points are used to specify the new location.

Using the Command

To use the command, select the 2-D entity or group of entities to be copied. Choose the FOUR-POINT MOVE command. Set four points for the new locations of the four corners of the selection. When the fourth point is set, the selection is moved.

See Also: *Four-Point Copy Command, Move Command*

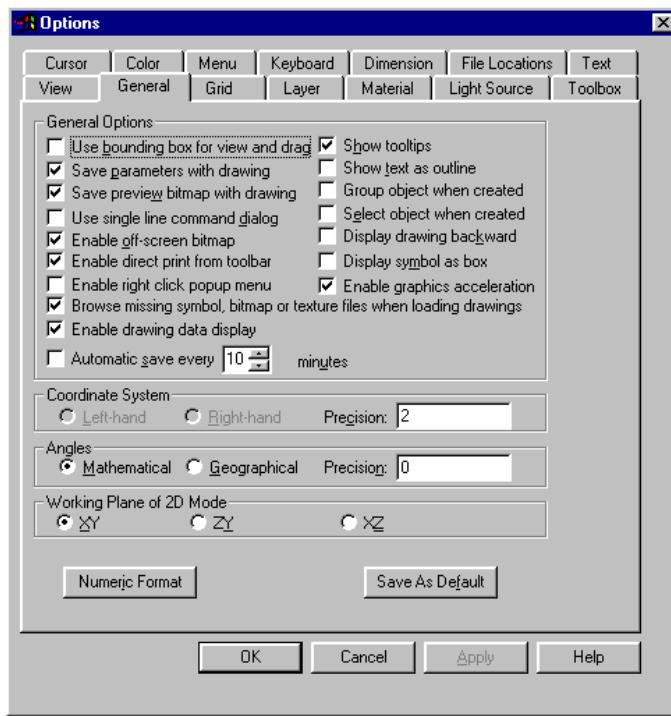
General Options

Menu:	OPTIONS
Menu Command:	OPTIONS

The General Options folder allows you to control general as well as coordinate and angle options.

Using the Command

Choose the OPTIONS command, and then click on the GENERAL tab to bring up the General Options folder.



General Options

Use Bounding Box for View and Drag

This option determines whether a Bounding Box or a bitmap is displayed when you set the view or drag an object by its selection handle.

Save Parameters with Drawing

This option saves your drawing with the current environment features, such as cursor step size, grid size and display, and other parameters.

Save Preview Bitmap with Drawing

When this option is enabled, a bitmap for the Preview area of the Open dialog box is automatically saved along with the drawing.

Use Single Line Command Dialog

When this option is enabled, options for DesignCAD commands appear in the Command Line (the space directly below the Command Menu that is normally occupied by the Toolbar) instead of command dialog boxes.

Enable Off-Screen Bitmap

When this option is active, DesignCAD saves a snapshot of the screen every time you perform certain functions. This option should normally be enabled because it makes the operation of DesignCAD faster.

Enable Direct Print from Toolbar

The Enable Direct Print from Toolbar option affects the functionality of the Print icon in the Main Toolbar. The option works as a toggle to make the icon bring up the Print Command dialog box or automatically print the current view of the drawing.

Enable Right Click Popup Menu

The Enable Right Click Popup Menu option affects the right mouse button. When this option is checked, the right mouse button brings up a right-click menu. All of the options in the right-click menu are available in other places in the program.

When the Enable Right Click Popup Menu option is disabled, clicking the right mouse button executes the Gravity command. The Gravity command moves the cursor to the nearest point in the drawing and set a point there.

Browse Missing Symbol, Bitmap or Texture Files when Loading Drawings

This option will display a browse dialog if a symbol, bitmap or texture file cannot be found so that you will be able to locate it.

Enable Drawing Data Display

When this option is active, DesignCAD displays various forms of information when drawing commands are used.

Show Tooltips

This option displays a yellow text box beside the cursor when the cursor is placed over a tool in a toolbox.

Show Text as Outline

This option shows just the outline of filled text entities.

Group Object When Created

When this option is enabled objects that are the result of an array, circular array, etc. are automatically grouped when they are drawn.

Select Object When Created

When this option is enabled, an object is automatically selected when it is drawn.

Display Drawing Backward

When this option is enabled, DesignCAD regenerates objects in the drawing in the reverse order of that in which they were created.

Draw Symbol as Box

The Draw Symbol as Box option represents symbols in drawings with boxes instead of the symbol. This feature greatly reduces redraw time of drawings containing several symbols.

Enable Graphics Acceleration

This option checks the Enable Graphics Acceleration options in the dialog boxes for the Shading and Hidden Line Removal commands by default.

Automatic Save

The Automatic Save Option saves a drawing automatically at intervals defined in the MINUTES box. An Automatic Save stores the open drawing with a name "*filename*(AutoSave).dcd." "*Filename*" is the name of the original drawing. The original file ("*filename.dcd*") is not modified until the user issues either the Save or Save As command. A Save or Save As updates the "*filename.dcd*."

If the file is saved and the program terminates normally, "*filename.dcd*" is updated and "*filename*(AutoSave).dcd" is removed. If the program crashes, the "*filename*(AutoSave).dcd" file won't be removed, so you can open "*filename*(AutoSave).dcd" to recover the drawing as it was up to the last Automatic Save.

Also, if you make a mistake and your drawing is "autosaved," you can close the program without saving the drawing file. The "*filename*(AutoSave).dcd" file is removed and the "*filename.dcd*" is unchanged from the last Save or Save As command.

Coordinate System

Left-hand

Choosing this option makes positive direction of the Z axis extend away from you.

Right-hand

Choosing this option makes positive direction of the Z axis move toward you.

Precision

Enter the number of digits that you want displayed to the right of the decimal in coordinate measurement.

Angles

Mathematical

With MATHEMATICAL angles selected, 0 degrees is the positive X axis (three o'clock) and the degrees progress counterclockwise.

Geographical

With GEOGRAPHICAL angles selected, 0 degrees is on the positive Y axis (twelve o'clock) and the degrees progress clockwise.

Precision

Enter the number of digits that you want displayed to the right of the decimal in angular measurement.

Working Plane of 2-D Mode

Click to select the workplane DesignCAD will display when in 2-D Mode.

Numeric Format

Click this button to open the Numeric Format dialog box. This dialog box contains options that control the format of coordinate and angular values displayed in the Coordinate Bar.

Coordinate Format

- decimal
- fractional
- engineering
- architectural

Angle Format

- degrees
- grads
- radians
- degrees, minutes, seconds

Save As Default Option

If you want to save the changes to the next session, click the **SAVE AS DEFAULT** button. Click **OK** when you are finished.

Gravity Command

Menu:	POINT
Menu Command:	GRAVITY
Shortcut Key:	. (period)
Mouse Shortcut:	Right Mouse Button
Toolbox Icon:	

This very useful command moves the cursor to the nearest point in the drawing and sets a point there. It allows you to set a point exactly on another point without having to "zero in" on it.

Using the Command

When you choose the command and you're in 3-D Selection Mode, the cursor moves to the nearest point in 3-D space. In 2-D Selection Mode, the cursor moves to the nearest point on the screen and does not take into account the point's location in 3-D space.

This is an important difference. If you want to be able to snap to the nearest point on the screen, you should use the 2-D Selection Mode. However, in 2-D Selection Mode, the cursor may seem to move in an arbitrary manner along the Z axis when you snap to a point because the cursor takes on the Z value of the point that it snaps to.

In 3-D Selection Mode, the cursor may seem to jump to a point other than the nearest point on the screen. This is because it goes to the nearest point in the drawing, in three-dimensional space. That point may not be the point that appears nearest on the screen.

Example: Set a point exactly on the right endpoint of a line.

Select the GRAVITY command. Move the cursor to a point near the right end of the line. Click the mouse button. The cursor snaps to the right endpoint and sets a point.

Gravity Move Command

Menu:	POINT
Menu Command:	GRAVITY MOVE
Shortcut Key:	, (comma)

The Gravity Move command snaps the cursor to the nearest point in the drawing, but it does *not* set a point there.

Using the Command

The Gravity Move command lets you move the cursor to another point precisely without having to "zero in" on it. As soon as you choose the command, the cursor snaps to the nearest point in the drawing without setting a point.

Gravity Move is often convenient for moving the cursor to a point in the drawing to get to a known location before using the cursor keys or the Point Relative command.

In 3-D Selection Mode, the cursor moves to the nearest point in 3-D space. In 2-D Selection Mode, the cursor moves to the nearest point on the screen and doesn't take into account the point's location in 3-D space.

This is an important difference. If you want to be able to snap to the nearest point on the screen, you should use the 2-D Selection Mode. However, in 2-D Selection Mode, the cursor may seem move in an arbitrary manner along the Z axis when you snap to a point because the cursor takes on the Z value of the point that is snapped to.

In 3-D Selection Mode, the cursor may seem to jump to a point other than the nearest point on the screen. This is because it goes to the nearest point in the drawing, in three-dimensional space. That point may not be the point that appears nearest on the screen.

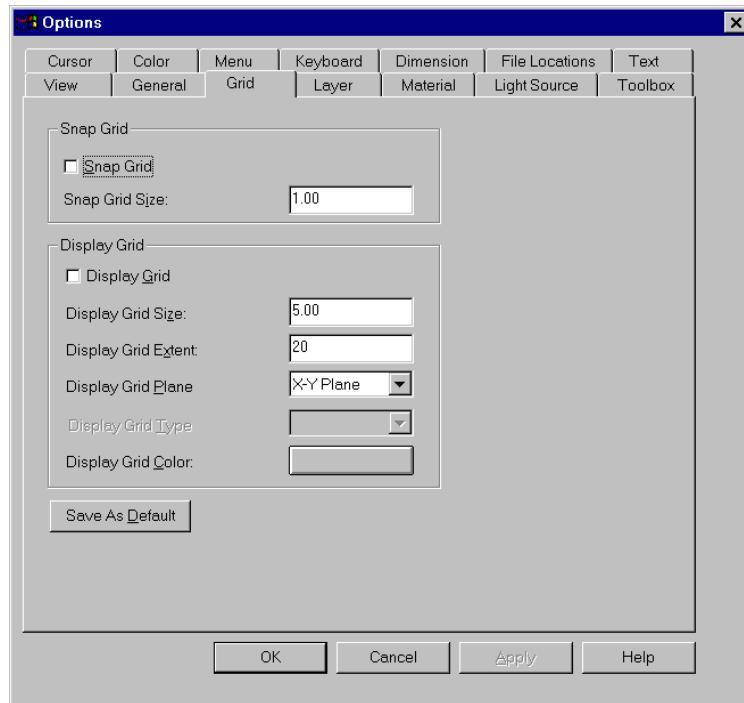
Grid Options

Menu:	OPTIONS
Menu Command:	OPTIONS
Shortcut Key:	Q

The Grid Options folder lets you set Snap and Display grid preferences.

Using the Command

Choose the OPTIONS command, and then click on the GRID tab to bring up the GRID OPTIONS folder.



Snap Grid

Snap Grid

Choosing this option forces the cursor to the nearest point on an invisible grid each time you set a point in the drawing.

Snap Grid Size

The size of the snap grid, in Drawing Units, can be set by clicking on the Snap Grid Size text box.

Display Grid

Display Grid

Clicking this checkbox forms a *visible* grid on the drawing screen.

Display Grid Size

Enter the number of Drawing Units desired for the height and width of each Grid Unit.

Display Grid Extent

Enter the number of Grid Units you want along the axes in each quadrant. This option only has an affect if DesignCAD is in 3-D Mode. In 2-D Mode the Display Grid will always extend to the edges of the drawing area.

Display Grid Plane

Choose the plane in which you want the grid to lie, by clicking on the down arrow beside the text box and then double clicking on the desired option. This option is only available in 3-D Mode. In 2-D Mode the plane for the Display Grid will always be the X-Y plane.

Display Grid Type

Choose the line type for the grid, by clicking on the down arrow and selecting it from the list box. This option is only available in 2-D Mode.

Display Grid Color

Selecting this option brings up the color palette. Choose which color you want the grid to be by clicking on that color in the palette.

Grid Settings Command

Menu:	OPTIONS
Menu Command:	GRID SETTINGS
Shortcut Key:	Ctrl+G

The Grid Settings command opens the Grid Options folder. The Grid Options folder contains options for both the Display Grid and the Snap Grid. The Grid sizes are measured in Drawing Units.

Using the Command

Choose the GRID SETTINGS command. The Grid Options folder appears. Set the options to meet your drawing needs. (See "Grid Options" for details.)

See Also: *Display Grid Command, Set Grid Center Command, Snap Grid Command, Units Command*

Group Define Command

Menu:	TOOLS
Menu Command:	GROUP DEFINE

The Group Define command puts all of the currently selected drawing entities into one group. The next time you select any part of the group, the entire group is selected. You can have many different groups defined in a drawing. Group Define makes it easy to keep related items together for copying, moving, scaling, and other operations.

Using the Command

Select all the objects you want to include in a group. When they are selected, choose the GROUP DEFINE command. DesignCAD treats the objects as a group and continues that way until the objects are ungrouped with the Group Explode command.

Example: Convert several objects into a single object.

Select the objects you want to redefine. Choose the GROUP DEFINE command. All of the objects will be recognized by DesignCAD as a single group.

See Also: *Group Explode Command*

Group Explode Command

Menu: TOOLS
 Menu Command: GROUP EXPLODE

The Group Explode command dissolves the currently selected group so that its members become single entities and can again be selected individually. Drawing entities are not affected except that they lose their association with the group.

Using the Command

With the group selected, choose the GROUP EXPLODE command. Undo does not cancel the effect of Group Explode. To restore group status, select all of the members again and use the Group Define command.

Example: Break up a group of objects into individual entities.

Select the group and choose the GROUP EXPLODE command. The group is redefined and the objects can be manipulated individually.

See Also: *Group Define Command*

Hammer Command

Menu: EDIT
 Menu Command: HAMMER
 Point 1: Source point on the grid (point where the hammer hits)
 Point 2: Destination point (where the hammer forces the grid)

The Hammer command can be used to reshape a surface grid.

Using the Command

Choose the Hammer command. In the dialog box choose the options you want. The size of the area affected by this command can be specified by entering the radius in the RADIUS box in the dialog box. Clicking in the FREE EDGES box lets the command warp the edges of the surface that is being hammered. You can also select whether the modified portion of the surface is to have a rounded or sharp point.

When you have specified the options you want, set a point on the surface grid at the location it is to be modified, and a point for the destination of that location. The surface is "hammered."

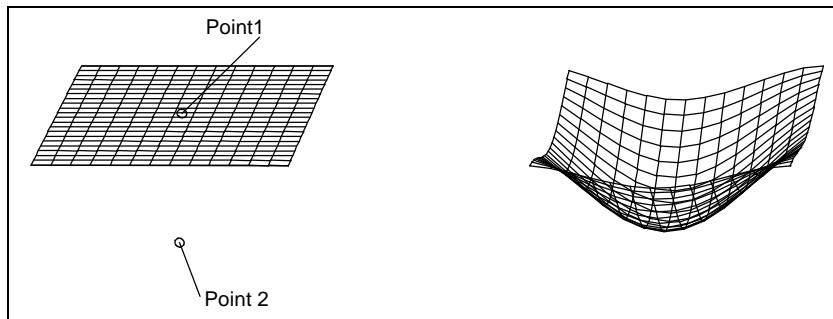


The Hammer command only moves existing points (intersections) on the grid. It does not create any new points. When you construct a grid to be modified with the Hammer command, you should make the spacing dense enough to get a smooth result.

Note: The Hammer command only affects grids created with the Surface Connect or Surface Patch commands. It does not affect planes, lines, curves, arcs, or extruded objects. It can be useful in creating specially shaped surfaces.

Example: Reshape a flat grid surface.

Choose the HAMMER command. Enter 20 in the RADIUS field and click the ROUNDED radio button. Make sure the FREE EDGES option is not enabled. Set a point on the center of the grid. Move the cursor outward from the center of the grid and set the second point. The grid will have a rounded impression in it.



Hatch Command

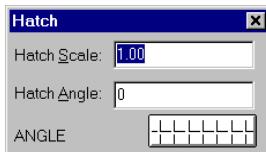
Menu:	DRAW
Submenu:	HATCH
Menu Command:	HATCH
Shortcut Key:	#
Points 1-n:	Outline of area to be hatched

The Hatch command fills an area with a hatch pattern. Points are set around the outside of the area to be hatched.

Using the Command

Choose the HATCH command from HATCH submenu in the DRAW menu.

Enter the size of the hatch pattern in the HATCH SCALE box in the dialog box. Enter the angle of the hatch pattern in the HATCH ANGLE box. Next, click the HATCH PATTERN button to choose the hatch pattern. Then set a point on the hatch pattern in the drawing.



Set points around the area to be hatched. Click the mouse or press **Enter**. The area is filled with the hatch pattern.

Note: The lines making up hatch patterns are defined as a single entity. These lines can be edited or erased only as an entity.

Scale and Angle

The scale and angle of the hatch patterns can be changed in the dialog box boxes. The scale is the size of the pattern elements. The angle is the angle at which the pattern is drawn.

Changing the Hatch Pattern

Click on the HATCH PATTERN button. The Hatch Pattern box appears. Click on the pattern style you want, then click the OK button or press **Enter**.

To change the scale of the preview on the Hatch Pattern button, press and hold either the **Shift** or **Ctrl** key while you click on the HATCH PATTERN button. The Shift key increases the scale of the preview. The Ctrl key decreases the scale of the preview.

Example: Draw an object filled with a brick pattern.

Select the HATCH command from the HATCH submenu of the DRAW menu. In the dialog box, set the HATCH SCALE, the HATCH ANGLE, and the HATCH FILL pattern. Return to the drawing and draw the object. Then press **Enter** to end the drawing command. The box is filled with the brick pattern.

See Also: **Hatch Fill Command, Hatch Line Command**

Hatch Fill Command

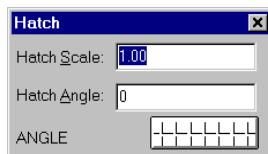
Menu:	DRAW
Submenu:	HATCH
Menu Command:	HATCH FILL
Point 1:	In the area to be hatched

The Hatch Fill command fills an area enclosed by lines with the specified hatch pattern.

Using the Command

Choose HATCH FILL from the HATCH submenu of the DRAW menu. Enter the size of the hatch pattern in the HATCH SCALE box in the dialog box. Enter the angle of the hatch pattern in the HATCH ANGLE. Click the HATCH PATTERN button to choose a hatch pattern.

To change the scale of the preview on the Hatch Pattern button, press and hold either the **Shift** or **Ctrl** key while you click on the HATCH PATTERN button. The Shift key increases the scale of the preview. The Ctrl key decreases the scale of the preview.



Next, set a point inside the area to be hatched. The area is filled automatically with the hatch pattern.

Note: The Hatch Fill and Hatch Line commands are only available if the program is in 2-D Mode.

See Also: *Hatch Command, Hatch Line Command*

Hatch Line Command

Menu: DRAW

Submenu: HATCH

Menu Command: HATCH LINE

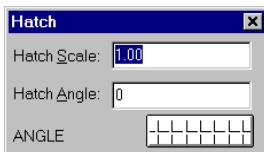
Point 1: On the enclosed line to be hatched

The Hatch Line command fills the area of an enclosed line with a hatch pattern. First, the hatch pattern is selected, and then points are set on the line or lines to be hatched.

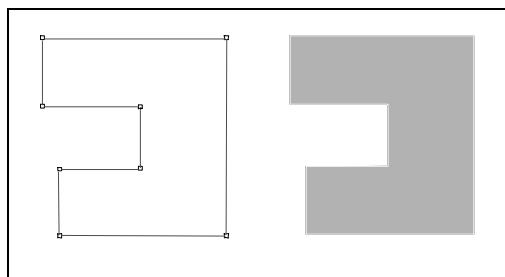
Using the Command

Choose the HATCH LINE command from the HATCH submenu of the DRAW menu. Enter the size of the hatch pattern in the HATCH SCALE box in the dialog box. Enter the angle of the hatch pattern in the HATCH ANGLE box. Click the HATCH PATTERN button to choose the hatch pattern.

To change the scale of the preview on the Hatch Pattern button, press and hold either the **Shift** or **Ctrl** key while you click on the HATCH PATTERN button. The Shift key increases the scale of the preview. The Ctrl key decreases the scale of the preview.



Next, select the line or lines to be hatched. Click the mouse or press **Enter**. The area inside the enclosed line is filled with the hatch pattern.



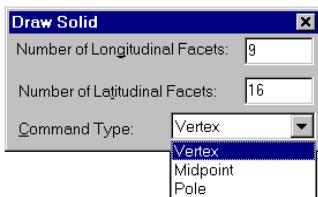
Note: The Hatch Fill and Hatch Line commands are only available if the program is in 2-D Mode.

See Also: **Hatch Command**, **Hatch Fill Command**

Hemisphere Command

Menu: SOLIDS
 Menu Command: HEMISPHERE
 Toolbox Icon: 
 Point 1: Center of the flat face
 Point 2: Radius of the flat face
 Point 3: Direction of the dome

The Hemisphere command draws solid hemisphere.



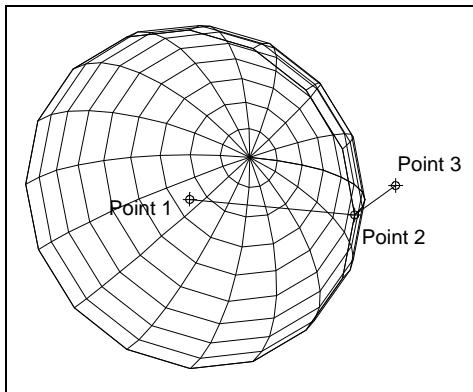
Using the Command

Choose the HEMISPHERE command. Enter in the NUMBER OF LONGITUDINAL FACETS and NUMBER OF LATITUDINAL FACETS boxes in the dialog box the number of sides you want the hemisphere to have from “equator” to “pole” and around its edge. You may also choose whether the second point represents a point at a vertex of the rim, or a midpoint of one of the flats on the rim.

When you draw the hemisphere, you may find it helpful to think of the object as a covered bowl. Point 1 is set for the center of the cover. Point 2 is somewhere on the rim. Point 3 defines the direction of the bottom of the bowl from Point 1.

Example: Add a hemisphere to your drawing.

Select the HEMISPHERE command and set a point for the center of the flat face. Move the cursor away from the first point. A rubber-band hemisphere will be drawn to represent the hemisphere using the cursor location as the second point. When the hemisphere is the desired radius, set the second point. Move the cursor to position the dome and set the third point.



Hidden Edge Command

Menu: EDIT
 Menu Command: HIDDEN EDGE

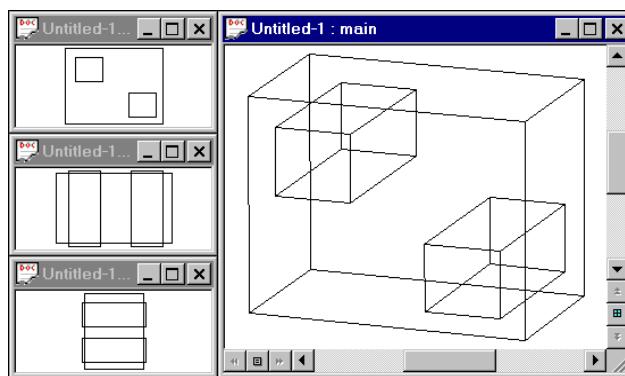
The Hidden Edge command can be used to erase cut-plane lines which are sometimes a result of the Cut Plane, Plane Subtract, and Solid Subtract commands.

Using the Command

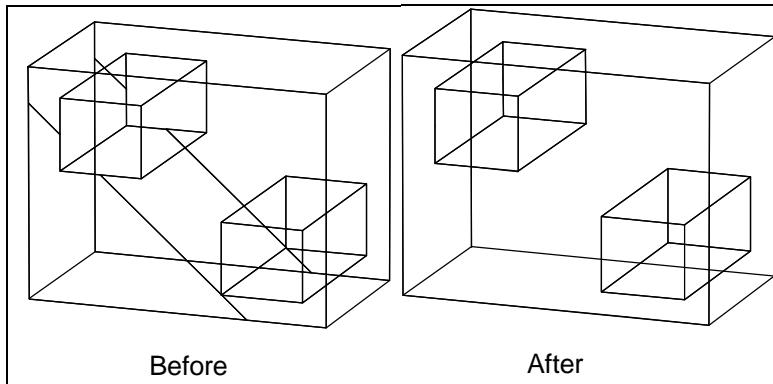
After executing a Cut Plane, Plane Subtract, or Solid Subtract command, select the HIDDEN EDGE command from the EDIT menu. Set a point on one of the cut-plane lines so that it will be hidden from the view or from a printout of the drawing.

Example: Subtract two boxes from another, then use the Hidden Edge command to hide any cut-plane lines.

Draw a large box and two smaller boxes. Draw the smaller boxes so that they are contained by the larger box with respect to the XY plane but intersect both ends of the larger box with respect to the YZ plane.



Use the SOLID SUBTRACT command to subtract the smaller boxes from the larger box. Notice the cut-plane lines which are a necessity of this kind of operation. Choose HIDDEN EDGE from the EDIT menu. Set a point on one of the cut-plane lines. The cut-plane line will be removed from the view and will not be printed in subsequent printouts. Repeat the Hidden Edge command as many times as desired.



Hidden Edge by Section Command

Menu: EDIT

Menu Command: HIDDEN EDGE BY SECTION

The Hidden Edge by Section command can be used to erase cut-plane lines which are sometimes a result of the Cut Plane, Plane Subtract, and Solid Subtract commands.

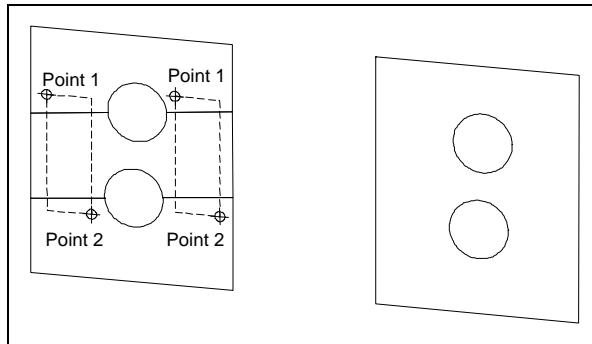
Using the Command

After executing a Cut Plane, Plane Subtract, or Solid Subtract command, select the HIDDEN EDGE BY SECTION command from the EDIT menu. Set two points to define a bounding box that encloses or touches the cut-plane lines, so that they will be hidden from the view or from a printout of the drawing.

Example: Subtract two circular planes from a rectangular one, then use the Hidden Edge by Section command to hide the cut-plane lines.

Draw a rectangular plane. Draw two circular planes in the same XY plane, within the boundaries of the rectangular plane. Use the Plane Subtract command to subtract the circular planes from the rectangular one.

Notice the cut-plane lines which are a necessity of this kind of operation. Choose HIDDEN EDGE BY SECTION from the EDIT menu. Set a point at one corner of the area that contains the cut-plane lines to the left of the circular holes. This point also defines one of the corners for a bounding (or rubber-band) box that moves with the cursor. Move the cursor so that the bounding box touches both of the edges to the left of the holes. Set another point. Both of the cut-plane lines to the left of the holes will be removed from the view and will not be printed in subsequent printouts. Now do the cut-plane lines on the right side of the holes.



Hidden Line Removal Command

Menu: **TOOLS**

Menu Command: **HIDDEN LINE REMOVAL**

Shortcut Key: **Ctrl+F8**

Toolbox Icon: 

The Hidden Line Removal command performs hidden line removal on the entire drawing or on a section of the drawing. This causes all lines behind surfaces to be removed, giving the objects a more realistic image.

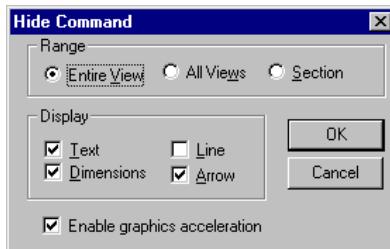
Using the Command

Choose the HIDDEN LINE REMOVAL command. Specify whether you want to remove lines for the entire drawing or only a section. If you choose the Section option, set two points to enclose the section on which you want DesignCAD to execute the Hidden Line Removal command.

You can also specify whether you want the text and dimensions to be displayed with the resulting image.

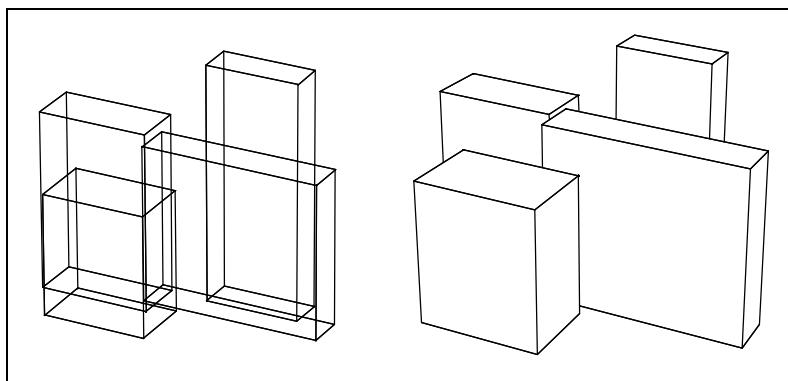
If your graphics card supports graphics acceleration, check the ENABLE GRAPHICS ACCELERATION option to improve performance. This will enable interactive hidden line removal. Interactive hidden line removal will allow you to move and edit existing drawing items and even add new ones, all while the specified view(s) maintain hidden line removal. The view(s) will remain in hidden line removal mode until you change the status of the view(s) to wireframe or shading.

If your graphics card supports graphics acceleration, you can also check the DISPLAY INTERSECTION LINES WITH GRAPHICS ACCELERATION ENABLED option to display lines at the intersections of solids and planes.



Example: Get a clear view of several objects arranged in front of and behind one another.

Select the HIDDEN LINE REMOVAL command. Choose the ENTIRE VIEW option and click OK. After the view has been redrawn, notice how much easier it is to see the orientation of the objects.



Import Command

Menu:	FILE
Menu Command:	IMPORT

The Import command lets you import files in several formats.

Using the Command

Choose the IMPORT command from the FILE menu. The IMPORT box appears. In the FILE NAME box enter the name of the file to import. In the LOOK IN box, tell the program where the drawing to import is stored. In the FILES OF TYPE box, enter the format of the file you want to import. When you have entered the information, choose the OPEN button to import the drawing. Choose the CANCEL button to return to the current drawing without importing a file.

DesignCAD 3D MAX imports the following file formats:

DWG

DesignCAD imports AutoCAD's DWG format directly.

DXF

DXF files can be used with many other Windows applications.

HPGL

HPGL is the Hewlett Packard graphics language. You can configure other applications for an HP plotter, send the plotter output to disk, and import that file into DesignCAD 3D MAX.

IGES

The IGES format is a standard format that many CAD systems support. DesignCAD 3D MAX supports the following IGES entities for input:

100 Circular Arc
102 Composite Curve
104 Conic Arc
106 Copious Data
108 Plane
110 Line
112 Parametric Spline Curve
114 Parametric Spline Surface
118 Ruled Surface
120 Surface Revolution
122 Tabular Cylinder
124 Transformation Matrix
212 General Note
214 Arrow
318 Define Subfigure
408 Insert Subfigure
412 Rectangular Array

Metafile

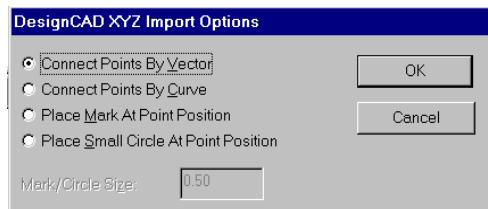
Many Windows applications can read Windows Metafiles, and the Import command allows you to bring those files into DesignCAD.

XYZ (or XY)

DesignCAD 3D MAX can import text files containing X,Y or X,Y,Z coordinates. The following conditions apply to the Import XYZ command:

1. The coordinates for each point should be on one line. They must be separated by either a space, a comma, a semi-colon, or a tab.
2. Individual line or curve entities must be separated by at least one empty line in the text file.
3. Any line which starts with a semi-colon is treated as a comment.
4. Comments may appear anywhere in the file.

You can connect the points with straight lines or smooth curves, or you can mark their position with a plus sign or small circle. If you choose to mark the points, you can set the size of the mark or circle in the edit box. The size is set in Drawing Units.



Example: Import an HPGL file.

Choose the IMPORT command. The IMPORT box appears. Designate HPGL format in the FILES OF TYPE box. Choose a path and file name for the file and click OK. The drawing is converted from HPGL file format and imported into DesignCAD.

Info Box Command

Menu:	VIEW
Menu Command:	INFO BOX
Shortcut Key:	Ctrl+I

The Info Box command can be used to view and edit properties of a selected object. To activate the command, you must select an object *before* executing the command.

Using the Command

Select an object and choose the INFO BOX command.

The available items in the Info Box vary depending on the type of object you have selected.

All Info Boxes contain Layer and Color information, a Same As button, and a Keep Visible button.



For details on the Layer and Color information, see the "Layer Options" and "Custom Color Command" entries respectively.

The Same As button lets you match the parameters of an existing entity. Just click on the SAME AS button, then click on the entity already in the drawing that has the properties you want for the selected entity.

The Keep Visible button controls whether or not the Info Box remains visible when no entity is selected. Click the KEEP VISIBLE button to set the Info Box visible when no entity is selected.

The button will appear to be "pushed in." To return the Info Box to its normal mode, click the KEEP VISIBLE button again. The button will pop back out.

If the Reverse Points button  is enabled (only when vector entities and/or spline curves are selected), click it to change the numbering of the points. The point designations are reordered; the first point becomes the last point and the last point becomes the first point.

If the Compliment Arc button  is enabled (only when arcs or elliptical arcs are selected), click it to change the arc to its complimenting arc.

If the Texture Mapping button  is enabled, click it to display the Texture Mapping dialog box. For more information on texture mapping, see the "Texture Mapping Command" entry.

If the Show Detail button  is enabled, click it to display additional details on the selected object like point locations, scale, etc.

There are six different kinds of Info Box configurations based on object categories. Each contains information relevant to the specific category of object.

Plane

- Point Coordinates: the coordinate of the current point on each axis.
- Current Point: the number of the point to which the information applies.
- Number of Points: the total number of points in the plane.
- Area: the area measurement of the plane in Drawing Units.
- Length: the measurement of the perimeter in Drawing Units.
- Material: the material properties of the entity.
- Smooth: True = Smooth enabled; False = Smooth Disabled.

Circle/Arc

- Center Coordinate: the coordinate of the center on each axis.
- Radius: the radius in Drawing Units.
- Start Angle: the starting angle of the Circle/Arc.
- Span Angle: the span angle of the Circle/Arc.
- Area: the area of the Circle/Arc.
- Length: the circumference of the circle or length of the arc in Drawing Units.
- Line Type: the type of line (i.e., dashed, dotted, etc.) used to draw the Circle/Arc.
- Line Scale: the amount of space between dashes or dots in certain line types.
- Line Width: the width of the line used to draw the Circle/Arc.
- Fill Wide Lines: fills wide line types in 2-D Mode instead of drawing them as outlines.

Dimension

- Text Font: the typeface of the text.
- Text Size: the point size of the text.
- Accuracy: the number of digits after the decimal point in the dimension.
- Arrow Type: the type of arrow used in the dimension.

- Format: the format of the dimension text.
- Flip Text: reverses the direction of dimension text.

Curve

- Point Coordinates: the coordinate of the current point on each axis.
- Current Point: the number of the point to which the information applies.
- Area: the area of the curve.
- Length: the measurement of the curve in Drawing Units.
- Line Type: the type of line used to draw the curve.
- Line Scale: the amount of space between dashes or dots in certain line types.
- Line Width: the width of the line used to draw the curve.
- Fill Wide Lines: fills wide line types in 2-D Mode instead of drawing them as outlines.

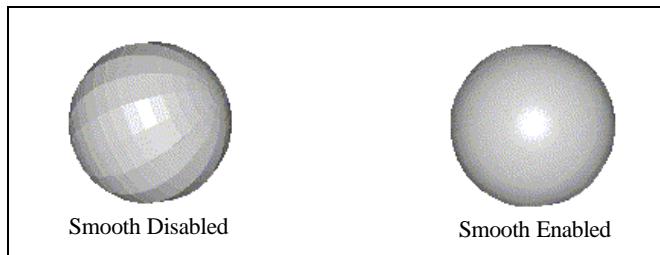
Text

- Font: the typeface of the text.
- Content: the letters, numbers, and symbols that make up the text entry.
- Angle: the angle at which the text entity is placed in the drawing.
- Length: the length of the text entity in Drawing Units.
- Height: the height of the text entity in Drawing Units.
- Keep Length/Width ratio: when checked the length/width ratio stays the same when the length or width is changed.

Solid

- X,Y,Z Scale: the scale of the solid along each axis.
- Material: the material properties of the solid.
- Smooth: Enables or disables the smooth shade option.

The Smooth shade option is selected by default for all solids. This causes curved surfaces to appear smooth, with their faceted edges rounded off. Smooth shading can be disabled in order to force the facets of the shaded solid to be visible.



Interference Check Command

Menu:	SOLIDS
Menu Command:	INTERFERENCE CHECK
Point 1:	First solid to check for interference
Point 2:	Second solid to check for interference

The Interference Check command checks two solids to see if they overlap one another at any point. To use this command, select the command and set a point on each solid. DesignCAD will display a dialog box stating whether or not interference is detected.



Example: Check to see if two solids intersect.

Select the INTERFERENCE CHECK command. Set one point on the first object and another on the second. DesignCAD will display the interference status. Click OK to remove the box from the screen.

Intersect-1 Command

Menu:	POINT
Menu Command:	INTERSECT-1
Shortcut Key:	F4
Toolbox Icon:	

Point 1: Line on which you want to find an intersection

The Intersect-1 command finds the nearest intersection to a point.

Using the Command

Set a point on the line on which you want to find the intersection. The cursor will move to the nearest intersection on that line and a point will be set there.

This command works on lines, circles, arcs, planes, and curves.

Example: Find the intersection of a line and a circle.

Select the INTERSECT-1 command. Set the cursor on the line and click the left mouse button. The cursor will move to the intersection of the line and the circle, and a point will be set there.

Intersect-2 Command

Menu:	POINT
Menu Command:	INTERSECT-2
Shortcut Key:	Shift+F4
Toolbox Icon:	

Point 1: First line of intersection
 Point 2: Second line of intersection

The Intersect-2 command locates the intersection of two entities, using two points as references.

Using the Command

Set a point on each entity. The cursor will move to the point at which they intersect.

This command works on lines, circles, arcs, planes, and curves.

Note: If the lines do not meet, then the cursor will move to a point where they would meet.

Example: Find where two lines intersect in a "web" of intersecting lines.

Choose the INTERSECT-2 command and set a point on one of the lines. Next set a point on one of the intersecting lines. The cursor will move to the intersection of the two lines.

Join Endpoints Command

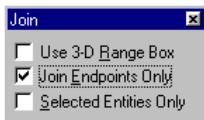
Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: JOIN ENDPOINTS
 Point 1: First corner of area to be joined
 Point 2: Second corner of area to be joined

The Join Endpoints command takes all of the points enclosed within a bounding box and moves them to a single point at the geometric center of the points. This command works with lines, and curves, as well as arcs drawn as lines.

Using the Command

Choose the JOIN ENDPOINTS command. Drag a box around the endpoints of the lines to be joined. Do not completely enclose all the lines or they will be reduced to a single, tiny point. The bounding box should enclose only the points of the lines you want to join, not the lines themselves.

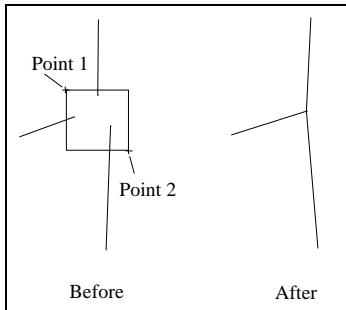
You can join only certain lines by selecting those lines before executing the command. Click in the SELECTED ENTITIES ONLY box and the selected lines will be joined. This makes it easy to join specific lines in a "busy" drawing.



To only join the endpoints of the lines, make sure that the JOIN ENDPOINTS ONLY box is checked. If the box is unchecked, all the points included in the bounding box will be joined.

Example: Join lines that do not meet at their endpoints.

Select the JOIN ENDPOINTS command. Set a point for one of the corners of the bounding box and move the cursor so that the rubber-band bounding box encloses all of the endpoints that you want joined. When you have done this, set the second point. The lines will be redrawn with their endpoints moved to the geometric center of the bounding box.



Keyboard Command

Menu:	TOOLS
Submenu:	CUSTOMIZE
Menu Command:	KEYBOARD

The Keyboard command is a shortcut method of bringing up the Keyboard Options folder of the Options file box.

Using the Command

Choose the KEYBOARD command from the CUSTOMIZE submenu in the TOOLS menu. The Options file box is displayed with the Keyboard Options folder showing. For a complete listing of the options available in this folder, see the "Keyboard Options" entry in the "Command Reference" section of this manual.

See Also: *Keyboard Options*

Keyboard Options

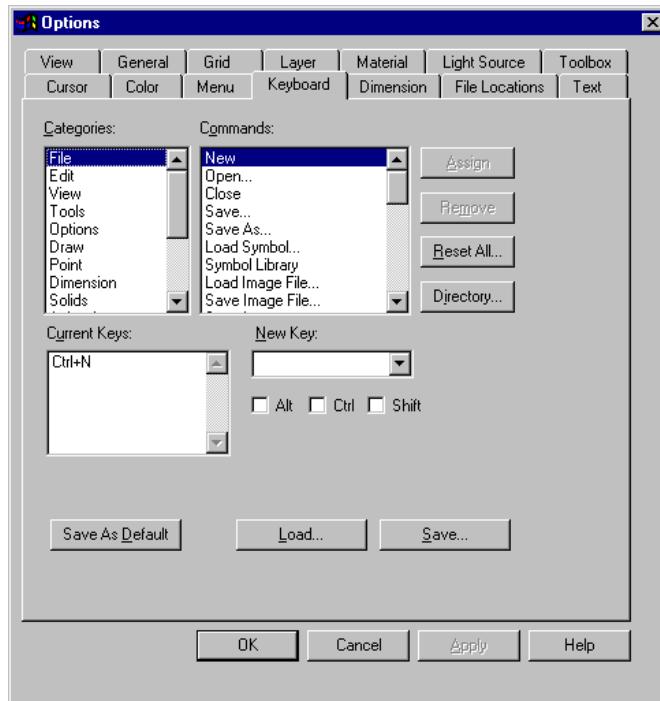
Menu:	OPTIONS
Menu Command:	OPTIONS

The Keyboard Options folder lets you change a shortcut key or assign one to any menu command, executable file or other application.

Shortcut keys let you go directly to a command from the keyboard. You can also use them to start an executable file — such as a Visual Basic program or another DesignCAD application — from within DesignCAD.

Using the Command

Choose the OPTIONS command from the OPTIONS menu. Click the KEYBOARD tab to make the Keyboard Options folder visible.



To find the command you're making a shortcut for, click the appropriate menu command in the CATEGORIES box. These correspond to the commands on the Main Menu.

Note: If you assign shortcut keys to applications in Windows, they may take precedence over the assignments in DesignCAD.

When a main menu command is selected in CATEGORIES, commands on its corresponding pull-down menu will appear in the COMMANDS box. Click a command name. If it has a shortcut assigned, the keys will appear in the CURRENT KEYS box.

To change or create a shortcut, click in the NEW KEY box, scroll through the keys list and click a key you want to assign. Check the ALT, CTRL or SHIFT box to select which function key you want to use with the other key. The new shortcut combination is now listed to the right of the New Key statement.

Click ASSIGN to finish making the shortcut, and the new shortcut appears in the CURRENT KEYS box. Repeat the steps if you want to create other shortcuts. Click the SAVE AS DEFAULT checkbox if you want DesignCAD to use these customized keyboard settings the next time you open the program. When the customization task is complete, click OK.

To load keyboard shortcut settings from an external file (*.dkf), click the LOAD button. Browse to the location of the shortcut settings file, select it, and click OPEN.

Click the SAVE button to save the keyboard shortcut settings as an external file (*.dkf). This file may then be used for one or more DesignCAD Workspaces.

To remove a shortcut key you've made, select the command you want to change, click on the shortcut in the CURRENT KEYS box, then click REMOVE. You can remove all shortcuts and return to default settings by clicking RESET ALL.

Example: Make a File Shortcut

Choose the OPTIONS command from the OPTIONS menu. Click the KEYBOARD tab. Scroll through the CATEGORIES box and click the desired file type. Select the drive and directory that contain the file to which you want to assign a keyboard shortcut. A list of files in that category and location appears in the FILES box. Click to select the desired executable file.

Click in the NEW KEY box, scroll through the keys list and click the key you want to assign. Check the ALT, CTRL or SHIFT box to select which function key you want to use with the other key. The new shortcut combination is now listed to the right of the NEW KEY statement.

Click ASSIGN to finish making the shortcut and the new shortcut appears in the CURRENT KEYS box. Repeat the steps to create other shortcuts. When you are finished, click OK.

See Also: *Options Command*

Layer Command

Menu:	OPTIONS
Menu Command:	LAYER
Shortcut Key:	L

The Layer command brings up the Layer Options folder, where you can enable or disable layers, name layers, select layers, and perform other functions.

Using the Command

Choose the LAYER command. The LAYER OPTIONS folder appears. Choose the settings you prefer and click the OK button. Press CANCEL to return to the drawing without keeping any changes.

See Also: *Layer Options*

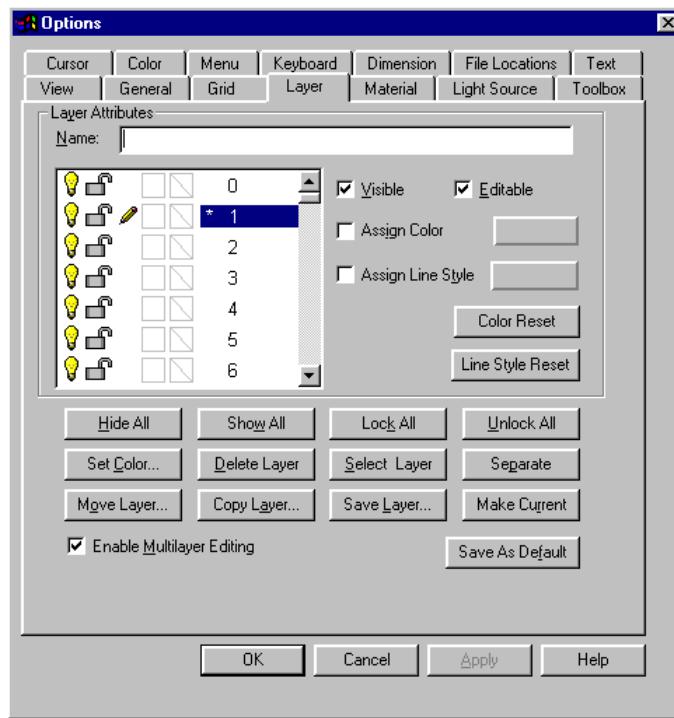
Layer Options

Menu:	OPTIONS
Menu Command:	OPTIONS
Shortcut Key:	Q

The Layer Options folder allows you to enable or disable layers, name layers, select layers, and perform other functions.

Using the Command

Choose the OPTIONS command from the OPTIONS menu and click on the LAYER tab. The LAYER OPTIONS folder appears. Choose the settings you prefer and click the OK button. Press CANCEL to return to the drawing without keeping any changes.



Layer Attributes

Name

You can assign names to the different layers in the drawing. To do this, highlight the layer number at the left and enter its new name in the NAME field.

To change the layer name using only the keyboard, press the **Tab** key until the focus is on a layer number. Use the up and down arrows on the keyboard to select the layer number for which you want to change the name. Press the **Spacebar**. Enter the new layer name and press **Enter**. Press **Esc** to cancel.

Editable

This option determines whether or not the objects in the highlighted layer may be edited. If a layer is not editable, it means that even though you can see the objects in the layer, you cannot modify them.

To change the editability status of a layer using only the keyboard, press the **Tab** key until the focus is on a layer number. Use the up and down arrows on the keyboard to select the layer number for which you want to change the editability status. Press the **Alt+E** to change the editability status of the layer.

Visible

You can hide layers by making them invisible. This is convenient when you need to reduce the "clutter" in a large drawing.

For example, you could hide all the electrical wiring in a house plan when you add the dimensions, or remove the text from a schematic when you add components.

To change the visibility status of a layer using only the keyboard, press the **Tab** key until the focus is on a layer number. Use the up and down arrows on the keyboard to select the layer number for which you want to change the visibility status. Press the **Alt+V** to change the visibility status of the layer.

Assign Color

When this option is enabled, you can assign a color to the currently selected layer. After a color is assigned to the layer all new items drawn on that layer will be drawn in the assigned color.

Assign Line Style

When this option is enabled, you can assign a line style to the currently selected layer. After a line style is assigned to the layer all new items drawn on that layer will be drawn using the assigned line style.

Color Reset

Clicking this button restores the assigned color for all layers to the DesignCAD default.

Line Style Reset

Clicking this button restores the assigned line style for all layers to the DesignCAD default.

Layer Commands

Hide All

This hides all layers in a drawing except the current layer.

Show All

This option makes all layers in a drawing visible.

Lock All

This makes all layers in drawing except the current layer uneditable.

Unlock All

This option makes it possible to edit all layers in a drawing.

Set Color

Assigns a color for all objects in a layer. All existing objects in the layer will be changed to the chosen color after the command is completed.

Delete Layer

This erases all entities in the highlighted layer.

Select Layer

This selects all objects in the highlighted layer.

Separate

This option sorts objects into layers by color. All objects of the same color will be put into the same layer, regardless of their previous layer.

Move Layer

This moves contents of one layer to another.

Copy Layer

This copies the contents of one layer to another.

Save Layer

This option saves the contents of an individual layer as a separate file.

Make Current

This option sets the selected layer as the active layer for the drawing.

Enable Multilayer Editing

Enabling Multilayer Editing allows you to manipulate all visible objects in editable layers. If this option is disabled, you can only manipulate objects in the active layer; all layers except the current layer are uneditable.

See Also: *Password Protection Manager Command*

Light Source Command

Menu:	TOOLS
Menu Command:	LIGHT SOURCE

The Light Source command brings up the Light Source Options folder, which gives you the option of setting up as many as eight different light sources. Each source is activated by clicking the checkbox beside it.

Using the Command

Choose the LIGHT SOURCE command. The Options file box is displayed with the Light Source Options folder showing. For a complete list of the options available in this folder, see Light Source Options.

See Also: *Light Source Options*

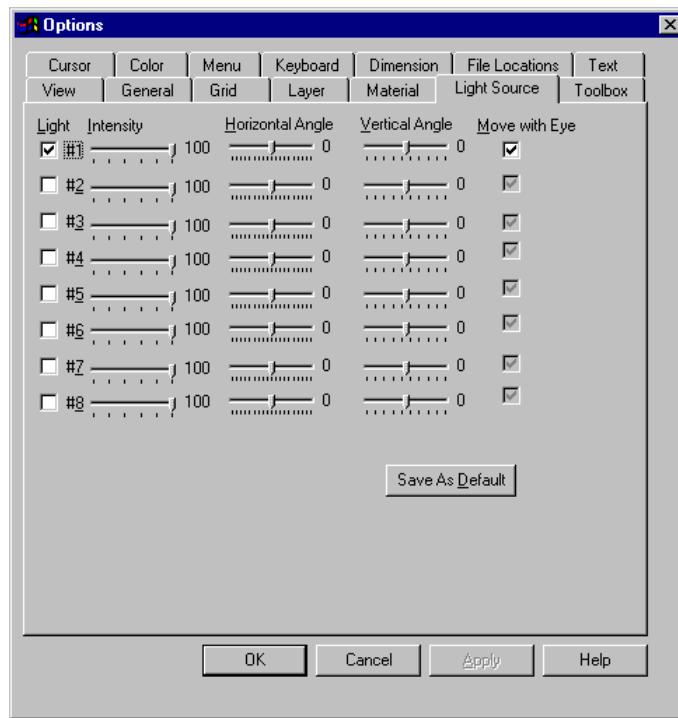
Light Source Options

Menu:	OPTIONS
Menu Command:	OPTIONS

The Light Source Options folder lets you set up as many as eight different light sources. Each source is activated by clicking the checkbox beside it.

Using the Command

Choose the LIGHT SOURCE command. Set the INTENSITY, HORIZONTAL ANGLE, and VERTICAL ANGLE. Then click the OK button.



Light Sources

Intensity

You can specify the intensity of each light source. Increasing the intensity of one light source does not increase the total amount of light. The total light is divided between each of the active light sources. If all eight light sources are turned on and set to intensity 100, each contributes 1/8 of the total light.

Horizontal Angle

This option can be used to set the horizontal angle of the light source relative to the drawn object.

Vertical Angle

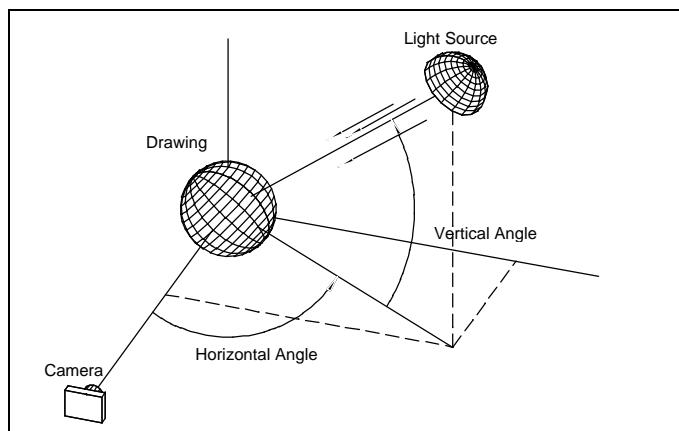
This can be used to set the vertical angle of the light source relative to the drawn object.

Move with Eye

When this option is enabled, the light sources move when the viewing angle changes. If the option is disabled, the light sources remain in their locations relative to the drawing when the viewing angle changes.

Example: Draw a sphere and smooth shade it.

Notice how the light reflects off of the object. Next, select the LIGHT SOURCE command. Set the VERTICAL ANGLE to **60** and the HORIZONTAL ANGLE to **45**. For this example, leave the other settings at their defaults, but be aware that you can change them to suit your preferences. Click OK. Now, reshade the sphere and notice the changes in appearance.



Line Command

Menu: DRAW

Submenu: LINES

Menu Command: LINE

Shortcut Key: **V**



Toolbox Icon:

Point 1-n: Points through which the line passes.

The Line command draws a two-dimensional or three-dimensional line. A valid line entity must contain two or more points.

Using the Command

Choose the Line command and set points for the line. You will see a rubber-band line connected to each point you set. When you have set the points, press **Enter** or double-click the mouse to end the command.

This command is similar to the Plane command except the Plane command creates a surface instead of a line.

Hint: You can use the Make Plane command to convert lines into planes so they can be shaded.

Example: Add a line to your drawing.

Select the LINE command. Set a point for the beginning of the line. Set several other points in different locations for the body of the line. Set a final point for the end of the line and press Enter.

See Also: **Make Plane Command, Ortho Line Command, Plane Command**

Line Connect Command

Menu: EDIT

Menu Command: LINE CONNECT

Point 1: Endpoint of the first line to be connected

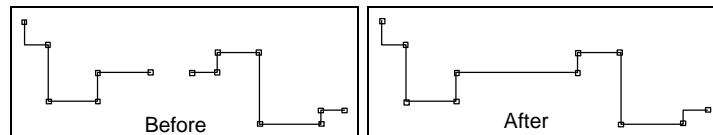
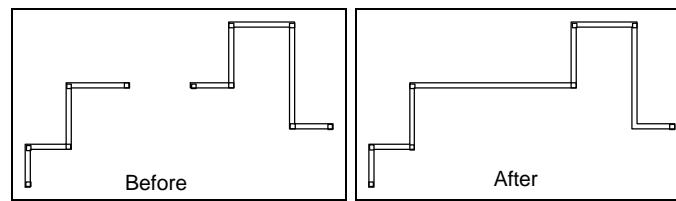
Point 2: Endpoint of the second line to be connected

The Line Connect command connects two lines or double lines.

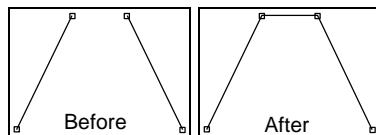
Using the Command

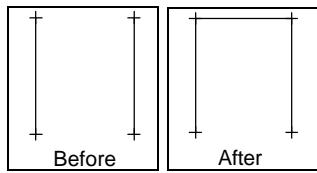
Select the LINE CONNECT command from the EDIT menu. Set a point on the endpoint of the first line or double line to be connected. Set a point on the endpoint of the second line or double line to be connected.

If the line segments to be connected are aligned, the gap between the two lines is seamlessly connected into a single line. The original endpoints of the combined segments are removed, so there are no points in the line to indicate the previous gap.



If the line segments are offset, DesignCAD adds an additional segment to connect the lines, resulting in a single line. In this case, the endpoints of the line segments are not removed.





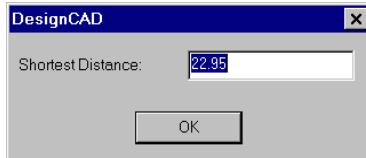
Line Distance Command

Menu: DIMENSION
 Submenu: INFO
 Menu Command: LINE DISTANCE
 Point 1: First line
 Point 2: Second line

The Line Distance command measures the shortest distance between two lines.

Using the Command

Set points on the lines to be measured. The shortest distance between the lines will be displayed on the screen.



Example: Find the shortest distance between two lines that do not intersect.

Select the LINE DISTANCE command and set a point on the first line. Set a point on the second line, and DesignCAD will display the shortest distance in a dialog box.

Line Plane Command

Menu: POINT
 Menu Command: LINE PLANE
 Point 1: Set a point on the intersecting line
 Point 2: Set a point on the intersecting plane

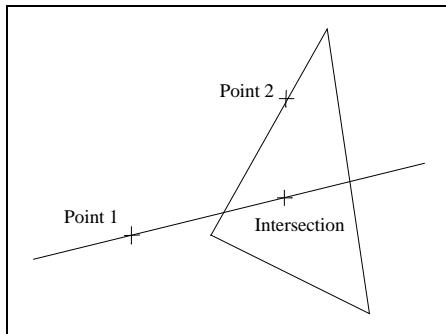
The Line Plane command sets a point at the intersection of a line and a plane.

Using the Command

Set one point on the line and a second point on the plane. If the line does not touch the plane, a point will be set where the intersection would be if the line were extended.

Example: Find the intersection of a line and a plane.

Select the LINE PLANE command. Set a point on the line and another on the plane. The cursor will move to the intersection.

**Line Snap Command**

Menu: POINT
 Menu Command: LINE SNAP
 Shortcut Key: K
 Toolbox Icon:

This command moves the cursor to the nearest line and sets a point there. It can be very useful in conjunction with drawing commands. For example, if you are drawing a line and want one of the endpoints to lie exactly on another line, you can use Line Snap to accomplish this without having to "zero in" on the line.

Using the Command

Choose the Line Snap command and set a point near the line to which you want to snap. The cursor snaps to the nearest point on the line and sets a point there.

Example: Set a point exactly on a line in your drawing.

Choose the LINE command again and then, before setting a point, choose the LINE SNAP command. Move the cursor close to the line and click the left mouse button. The cursor snaps to the line and sets the starting point of your second line.

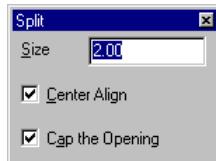
Line Split by Distance Command

Menu: EDIT
 Menu Command: LINE SPLIT BY DISTANCE
 Point 1: location of line split
 Point 2: direction of line split, if necessary

The Line Split by Distance command removes a section of a specified length from a line or double line, splitting the line or double line into two segments.

Using the Command

Select the SPLIT LINE BY DISTANCE command from the EDIT menu. The Split dialog box appears.

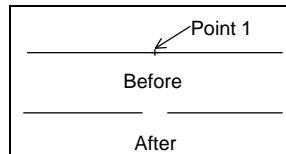


In the SIZE box, enter the desired length of the segment to be removed from the line or double line. If the CENTER ALIGN option is checked, set a point at the center of the segment to be removed. If the CENTER ALIGN option is not checked, set the first point for the location of the split and set a second point for the direction of the segment to be removed. Press **Enter** and the line is split by the distance entered in the Size box.

Note: DesignCAD will automatically cap the ends of a split double line if the CAP THE OPENING option is checked.

Example: Split a line by a distance of 2.

Draw a line. Select the SPLIT LINE BY DISTANCE command from the EDIT menu. Enter **2** in the SIZE box and make sure the CENTER ALIGN option is checked. Set a point near the center of the line. The line splits into two segments, with a gap 2 units long between them.



Line Split by Points Command

Menu: EDIT

Menu Command: LINE SPLIT BY POINTS

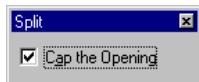
Point 1: first point of segment to be removed

Point 2: second point of segment to be removed

The Line Split by Points command removes a section from a line or double line, splitting the line or double line into two segments.

Using the Command

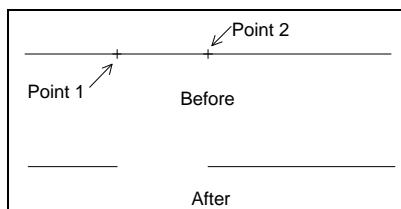
Select the LINE SPLIT BY POINTS command from the EDIT menu. DesignCAD will automatically cap the ends of a split double line if the CAP THE OPENING option is checked.



Set the first point for the segment to be removed. Set the second point for the desired length and direction of the segment to be removed. The portion of the line between the two points is deleted, splitting the original line into two segments.

Example: Split a line into two segments.

Draw a line. Choose the LINE SPLIT BY POINTS command from the EDIT menu. Set the first point for the line split. Set the second point for the line split. The area between the two points set is removed.



Line to Curve Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	LINE TO CURVE

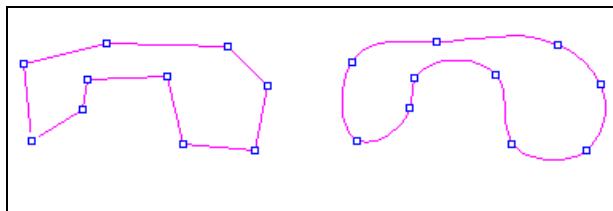
The Line To Curve command converts all selected lines to curves, using the vertices of the lines as defining points for the curves.

Using the Command

Select the line to convert. Run the LINE TO CURVE command. The line is converted to a curve.

Example: convert a closed line to a closed curve.

Draw a closed line, as shown, and select it. (The figure is shown in Point Select Mode to illustrate what happens to the points.) Choose the Line to Curve command. The line is converted to a curve. Notice in the figure that the points for the curve are in the same locations as for the line.



See Also: Curve to Line Command

Load Animation Template Command

Menu: ANIMATION
 Menu Command: LOAD ANIMATION TEMPLATE

The Load Animation Template command opens an animation template that has been produced in Animation Mode and then saved as a part of the drawing file with the Save Animation Template command.

Using the Command

After producing an animation in Animation Mode, save your work as an animation template so that it can be opened and viewed later. Select the SAVE ANIMATION TEMPLATE command from the ANIMATION menu. Enter a name for the animation in the dialog box and then click OK.

Note: The animation template is saved as a part of the drawing file it was created with and can only be opened later with the Load Animation Template command if the drawing file is open in DesignCAD.

See Also: *Animation Mode Command, Save Animation Template Command*

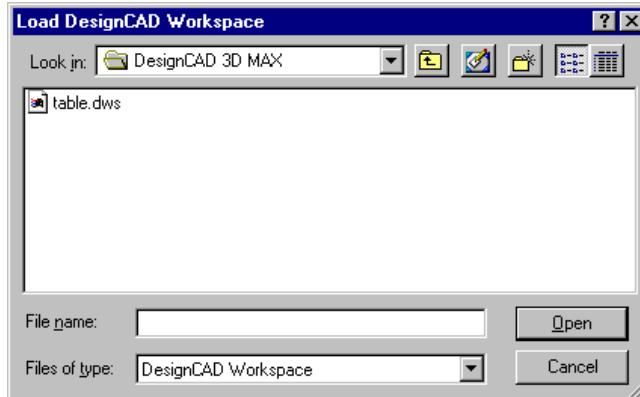
Load DesignCAD Workspace Command

Menu: FILE
 Submenu: WORKSPACE CONFIGURATION
 Menu Command: LOAD DESIGNCAD WORKSPACE

The Load DesignCAD Workspace command is used to load or change a DesignCAD Workspace.

Using the Command

Choose the LOAD DESIGNCAD WORKSPACE command from the WORKSPACE CONFIGURATION submenu of the FILE menu. The Load DesignCAD Workspace dialog box appears.



Select the template you wish to use and click OK. A dialog box appears to ask if you want to use the selected template as the default workspace template (when new drawings are started and in

subsequent drawing sessions). Click YES, NO, or CANCEL. If you click YES or NO DesignCAD is arranged according to the workspace you selected. If you click CANCEL, DesignCAD returns to the drawing without changing the workspace.

To change to a different workspace, follow the same steps. DesignCAD is rearranged according to the new template.

See Also: *Setup DesignCAD Workspace Command*

Load Digitizer Menu Command

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	LOAD DIGITIZER MENU

The Load Digitizer Menu command loads the "data" portion of a digitizer menu. The "paper" portion of the menu should be attached to the digitizer before the digitizer is loaded. A maximum of 10 digitizer menus can be loaded at once.

Using the Command

First, attach the paper template onto the digitizer. Choose the LOAD DIGITIZER MENU command from the DIGITIZER submenu of the TOOLS menu. The Load Digitizer Menu box appears. Choose the digitizer menu name in the FILE NAME box, then click OK or press **Enter**.

Now set a point in the lower-left corner of the digitizer menu. A rubber-band box shows how the digitizer menu will be loaded. Set a point in the upper-right corner of the digitizer menu.

See Also: *Create Digitizer Menu Command, Save Digitizer Menu Command, Add Menu Item Command*

Load Image File Command

Menu:	FILE
Submenu:	IMAGE
Menu Command:	LOAD IMAGE FILE
Point 1:	Lower-left corner for image placement
Point 2:	Upper-right corner for image placement

The Load Image File command adds a graphic image from another file to your drawing. If you only set a single point, the image will use that point as the lower-left corner and be retrieved at full scale. If you set two points, the image will be scaled to fill a rectangle with those two points as opposite corners. The image is never rotated in 3-D space; it always appears "flat" relative to the screen, regardless of your viewing angles.

Using the Command

Choose the LOAD IMAGE FILE command. Choose the type of file to be added to the drawing from the drop-down list in the FILES OF TYPE box. In the LOOK IN box enter the location of the bitmap. In the FILE NAME box enter the name of the file to load or select the file from the area below the Look In box.

This command is very useful in adding illustrations to your drawings. For example, it's possible to add a scanned photograph to be displayed along with your drawing.

Example: Insert a bitmap image into your drawing.

Select the LOAD IMAGE FILE command. The LOAD IMAGE FILE box appears. Select the desired .BMP file and click OK. Set a point in your drawing for one corner of the insertion box. Move the cursor away from the first point to stretch the rubber-band box. When the box is the correct size, set the second point. The bitmap image will be inserted into your drawing in the space formerly occupied by the insertion box.

Load Paper Space Template Command

Menu:	FILE
Submenu:	PAPER SPACE VIEW FRAME SETUP
Menu Command:	LOAD PAPER SPACE TEMPLATE

The Load Paper Space Template command is used to load or change a Paper Space Template.

Using the Command

Choose the LOAD PAPER SPACE TEMPLATE command from the PAPER SPACE VIEW FRAME SETUP submenu of the FILE menu. The Paper Space Template dialog box appears. Select the template you wish to use and click OK. To use a blank template, click OK without selecting one of the existing templates. Your drawing is arranged in the Paper Space according to the template you selected.

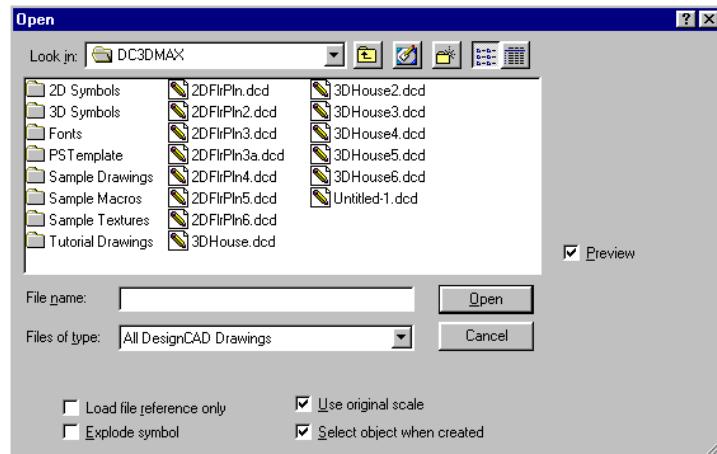
To change the template, follow the same steps. Your drawing is rearranged in the Paper Space according to the new template.

Load Symbol Command

Menu:	FILE
Menu Command:	LOAD SYMBOL
Shortcut Key:	Ctrl+F9
Point 1:	First handle for symbol placement
Point 2:	Second symbol handle (optional)
Point 3:	Third handle (optional, only used for a 3D drawing or symbol)

The Load Symbol command loads a drawing symbol or merges an existing drawing with the current drawing.

In DesignCAD, symbols inserted into the drawing with the Load Symbol command are defined as single entities, whether they are saved as part of the drawing or linked to the drawing. That is, the entire symbol is selected, moved, and erased as a single entity.



Load File Reference Only

This checkbox determines whether the drawing being added or "merged" with the current drawing will be saved as a part of the drawing or just linked by a reference.

If the option is not checked and the symbol is saved as part of the current drawing, the file size will be larger, and the drawing will not be updated when the symbol file is modified. The advantage is that the drawing will not be affected if the symbol file is deleted, or the drawing is opened on a computer that doesn't have a copy of the symbol file.

If the box is checked, the program adds an "insertion entity" to the drawing file. The symbol file is read every time the drawing is loaded. The symbol file must be present and in its original location. If the symbol file is modified, the change will be reflected in all the drawings using that symbol.

Explode

When a referenced symbol is exploded, the symbol file is not read from disk when the drawing is loaded. A copy of the symbol is placed into the drawing instead of an insertion entity, just like a symbol that was loaded without using the Load File Reference Only option.

Use Original Scale

If the Use Original Scale option is checked, the size of the symbol will be fixed and only the angle of the object will change when additional points are set to specify drawing handles.

Select Object When Created

When this option is enabled, the symbol is automatically selected when it is added to the drawing.

Using the Command

Choose LOAD SYMBOL from the FILE menu. The OPEN dialog box appears. From the file list, select the name of the symbol you want to load. Then click the OK button.

A rubber-band box appears in the drawing. This shows the area of the symbol. Set one to two points to establish the location, size, and orientation of the symbol. To accept the symbol's size and orientation, set a point to establish the location, and then press **Enter**.

Manipulating a Symbol

A symbol must be "exploded" in order to change its characteristics such as color, line style, etc. There are two ways to explode a symbol: the Explode option in the Load Symbol dialog box (before the symbol is loaded); or the Explode command in Edit | Selection Edit (after the symbol is loaded).

When a referenced symbol is exploded with the Explode option or command, the symbol file is not read from an external file when the drawing is loaded. A copy of the drawing information for the symbol is placed into the drawing instead of an insertion entity.

When a non-referenced symbol is exploded with the Explode option or command, there is no change to the drawing unless it contains multiple copies of the same symbol. If several, non-referenced copies of a symbol exist in the same drawing, they share drawing information even though that drawing information is contained in the drawing file that contains the symbols (not an external drawing file). In order to manipulate one of the copies of the symbol, it must be exploded so that it will have its own set of drawing information.

Once a symbol has been exploded, it will still be recognized as a group. This way it is not necessary to select many small items just to change the color, line style, etc. of the former symbol. The Group Explode command may then be used if individual items need to be manipulated.

Macro Execute Command

Menu:	TOOLS
Menu Command:	MACRO EXECUTE
Shortcut Key:	%

This command runs a DesignCAD macro. A macro is a saved set of drawing actions that can be used later as a single command.

Using the Command

After choosing the command, enter the macro name when asked for it, and then set a starting point for the macro. The macro begins its execution from that starting point.

Example: Draw a box of a specific size in several drawings.

Select the MACRO RECORD command. Name the macro TEST.D3M and click **SAVE**. Draw a box and a sphere on the screen. Click the stop icon. The macro will be saved under the selected file name. Then, select the MACRO EXECUTE command and choose **TEST.D3M** from the list box. Set a starting point for the macro and DesignCAD will carry out the recorded series of actions.

See Also: **Macro Record Command**

Macro Record Command

Menu: TOOLS
 Menu Command: MACRO RECORD

A macro is a saved set of drawing actions that can be used later as a single command.

Using the Command

When you use the Macro Record command, you are first asked for the macro name. Select a name and a location for the macro in the RECORD MACRO box. Then the Macro Record toolbox appears on the screen:



The icons represent RECORD/CONTINUE, PAUSE, MACRO OPTIONS, and STOP. The Macro Options icon brings up a dialog box in which you can set different options for the macro.



Record Layer

Checking this box causes the macro to run in the same layer in which it was created.

Record Command Parameters

Checking this box saves command bar information in the macro. For example, if you create a sphere with this option enabled, the macro will retain information such as number of latitudinal and longitudinal facets, and whether you created it in Vertex, Pole, or Midpoint format. If you leave this option disabled, then you will be asked for that information each time you run the macro.

Record Points

Checking this box causes the points set in specific drawing commands to be retained in the macro. This will affect dimensions of objects and their orientation to each other, but the absolute location of objects in the drawing is determined by the starting point that you select each time you run the macro.

Record Color

Checking this box configures the macro to create all objects in the same colors in which they were recorded. After you enter the macro name, everything you draw on the screen

becomes part of the macro. You can pause recording by clicking on the PAUSE button, and resume by clicking on the RECORD button.

When you are finished recording your macro, click the STOP icon or choose STOP RECORDING from the TOOLS menu. The macro can then be run using the Macro Execute command.

Example: Draw a box and a sphere of specific sizes in several drawings.

Select the MACRO RECORD command. Name the macro TEST.D3M and click SAVE. Draw a box and a sphere on the screen. Click the STOP icon. The macro will be saved under the selected file name. Then select the MACRO EXECUTE command and choose TEST.D3M from the list box. Set a starting point for the macro and DesignCAD will carry out the recorded series of actions.

See Also: Macro Execute Command

Macro Toolbox Command

Menu:	TOOLS
Menu Command:	MACRO TOOLBOX

This command is a toggle command that either displays or hides the Macro Toolbox, depending on the current status of the toolbox. The following macro options are available in the Macro Toolbox: Continue, Pause, Record Options, and Stop.

Using the Command

After selecting the MACRO RECORD command, bring up the Macro Toolbox by choosing the MACRO TOOLBOX command. To hide the toolbox choose the MACRO TOOLBOX command again. This command is not available until you choose the Macro Record command.



Make Plane Command

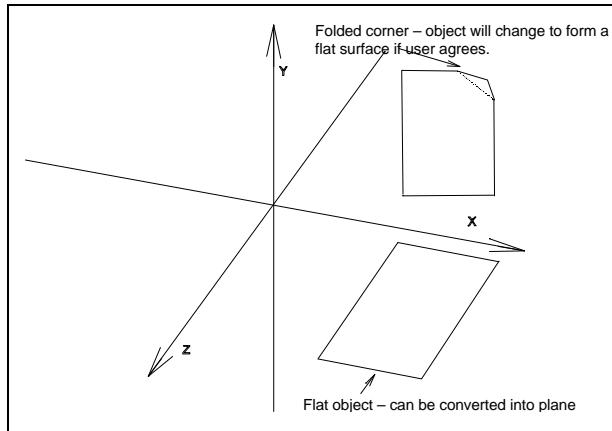
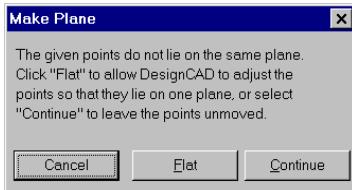
Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	MAKE PLANE

The Make Plane command can be used to convert lines, curves, circles, and arcs into a plane. Planes can be shaded, whereas lines, circles, and arcs cannot.

Using the Command

Select the line or lines to be converted to a plane. The selected lines should all form a flat surface.

If this condition is not met, DesignCAD brings up this box:



Example: Make a plane from three or four lines that meet at their endpoints and form a closed object (triangle, square, etc).

Select the lines as a group and choose the MAKE PLANE command. The lines will be converted into a plane surface.

Material Command

Menu:	OPTIONS
Menu Command:	MATERIAL

The Material command brings up the Material Options folder which can be used to specify the material properties of an object. You can choose the current material from the materials provided. In other words, you select the material in which the next object or objects will be drawn. You can also create your own materials.

After the material is selected, objects drawn will have that material property. For example, if you select the material Walnut and then draw a hemisphere, the hemisphere will look like Walnut wood. You can assign a material to an existing object with the Info Box.

Using the Command

Choose the MATERIAL command from the OPTIONS menu. The MATERIAL OPTIONS folder appears. Choose the settings you prefer and click the OK button. Press CANCEL to return to the drawing without keeping any changes.

See Also: Material Options

Material Options

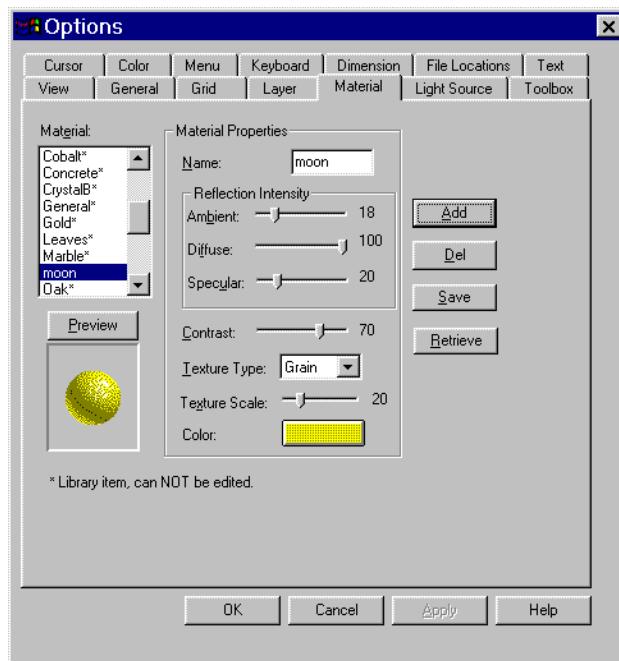
Menu: OPTIONS
 Menu Command: OPTIONS

The Material Options folder can be used to specify the material properties of an object. You can choose the current material from the materials provided. In other words, you select the material in which the next object or objects will be drawn. You can also create your own materials.

After the material is selected, objects drawn will have that material property. For example, if you select the material Walnut and then draw a hemisphere, the hemisphere will look like Walnut wood.

Using the Command

Choose the OPTIONS command and click on the MATERIAL tab. The Material Options folder appears. Make the selections to get the material you want, then click the OK button. Draw your object and shade it.



Preview

Displays a sphere composed of the selected material in smooth shade mode.

Material Properties

Name

Displays the name of the selected material. You can enter a name for new materials by clicking the box and entering the name you want.

Reflection Intensity**Ambient**

This determines the amount of background light shining on the object. This dictates how much shadow is visible on a shaded object.

Diffuse

Determines the amount of flat (not shiny) reflection you get from a shaded object.

Specular

Determines how shiny (not flat) the material appears.

Contrast

Determines how shiny the surface appears. The lower the setting, the shinier the surface appears.

Texture Type

Set the texture of the material as NONE (smooth), MARBLE, GRAIN, WOOD, CEMENT, PATCH.

Texture Scale

Determines the amount and size of the texturing.

Color

Displays the color of the selected material. Clicking on this bar will bring up the color palette, where you can specify a color for your material by clicking on a color tile.

Add

Creates a custom material. You can set preferences for all of the material's properties.

Del

Deletes a material from the materials selection box.

Save

Saves created materials as separate files.

Retrieve

Retrieves a specified material file and inserts it into the selection box.

Example: Create a new material.

Select the MATERIAL command and click the ADD button. Enter a name for your material in the NAME edit box. Enter values for the different material properties. Click the PREVIEW button to see what the material looks like with the current settings and play with the settings until the material is to your liking. When you have completed your material, click the SAVE button.

Material List Command

Menu:	FILE
Menu Command:	MATERIAL LIST

The Material List command brings up a list of all the attributes and their quantities in the current drawing. This list can be copied to the Clipboard, printed, or saved to a file.

Attributes generally refer to physical materials—lumber, bolts, screws, and other parts—used to build the object depicted in a drawing. The Material List can be used for cost estimating, parts lists, and other functions that require a list of items used in a drawing.

Using the Command

Choose the MATERIAL LIST command. The MATERIAL LIST box appears, showing a list of materials in your drawing and the count of each.

Note: The Material List command (in the File menu) should not be confused with the Material List in the Material Toolbox. "Material," as it is referred to in the Material Toolbox, affects the appearance of a shaded drawing. The Material List command provides a listing of Attributes placed in a drawing.

Example: Show a list of all the labeled parts (Attributes) in the object you are drawing.

Select the MATERIAL LIST command. The MATERIAL LIST box displays the names and quantities of all the Attributes in your drawing.

Menu Command

Menu:	TOOLS
Submenu:	CUSTOMIZE
Menu Command:	MENU

The Menu command is a shortcut method of bringing up the Menu Options folder in the Options file box.

Using the Command

Choose the MENU command from the CUSTOMIZE submenu in the TOOLS menu. The Options file box is displayed with the Menu Options folder showing. For a complete list of the options available in this folder, see Menu Options.

See Also: Menu Options

Menu Options

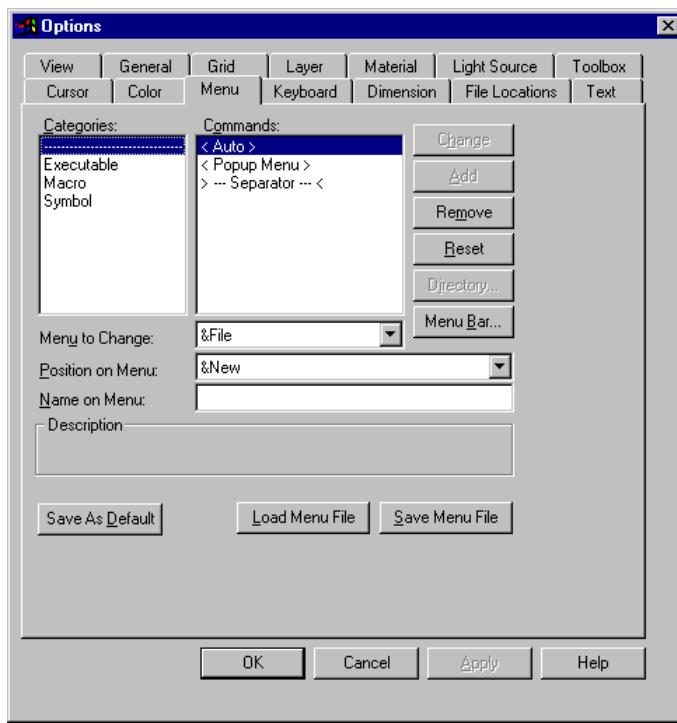
Menu:	OPTIONS
Submenu:	OPTIONS

The Menu Options folder allows you to customize how commands appear on menus. You can also add commands you create to any menu.

For example, you can group often-used commands on a particular pull-down menu or create a command with the BasicCAD programming language and add it to a menu. You can also record a macro and assign it to one of the pull-down menus.

Using the Command

Choose the OPTIONS command from the OPTIONS menu. Click on the MENU tab. The Menu Options dialog box comes up on the screen.



To find the menu where the command you wish to add is located, click the appropriate menu in the CATEGORIES box. These correspond to the commands on DesignCAD's main menu. When a main menu is selected in Categories, commands on its corresponding pull down menu will appear in the Commands box.

When you're looking for an executable file, a macro or a symbol in the lower three entries in the Categories box, the label on the Commands box changes to Files. The instructions below refer to the Commands box, but they also apply to the Files box. The box name changes depending on what category you're looking at, but the box's function remains the same.

Change

To change a command name, select the main menu in which the command you wish to change is located in the MENU TO CHANGE box. Select the command to change in POSITION ON MENU box. Click in the NAME ON MENU box, enter a new name, then click CHANGE and OK.

Add

When you add a command to a pull down menu, it's always inserted above the command that appears in the Position on Menu box.

In the CATEGORIES: box, select the main menu that contains the command you wish to add. Choose the command you wish to add to the menu from the list of possible commands in the COMMANDS: box. Select the main menu you want to add the command to in the MENU TO CHANGE box. Choose where you want to place the new command in the POSITION ON MENU box. (You'll want to select the command that will be below your new command.)

Click in the NAME ON MENU box, enter a name for the command. Click ADD and then click OK to finish modifying the menu.

Note: To add an executable file, you must click the DIRECTORY button to find the file's location on the hard drive.

Remove

To delete a command from a menu, scroll down and select the main menu in the MENU TO CHANGE box. Scroll down and select the command you want to remove in the POSITION ON MENU box. Click the REMOVE button, then click OK.

Reset

Return modified menus to their original DesignCAD defaults by clicking the RESET button and the OK button.

Menu Bar

You can add, remove or rename entire categories of commands from the main menu command bar by clicking the MENU BAR button, which brings up the Menu Bar dialog box.

For example, you could create a set of commands to handle certain symbols, and then add a command category named Symbols to the menu bar between the WINDOW and HELP commands.

To do this, enter the name of the new command category in the NAME ON MENU box, select where you want to place the command in the POSITION ON MENU box, then click the ADD button and click OK. You may now add specific pull down menu commands to the Symbols category by using the Add command.

Any change made to a main menu category will affect all its pull down menus. For instance, when you remove a main menu command, all its corresponding pull down menu commands are removed as well.

Save as Default

To keep the menu changes for the next time you run DesignCAD, you must check the SAVE AS DEFAULT box. If the box is not checked, your menu modifications will be in effect only for the current session and will be lost when you close DesignCAD.

Load Menu File

Click the LOAD MENU FILE button to load a previously saved menu file.

Save Menu File

Click the **SAVE MENU FILE** button to save the menu settings as an external file. This file may then be used for one or more DesignCAD Workspaces.

Midpoint Command

Menu:	POINT
Menu Command:	MIDPOINT
Shortcut Key:	Ctrl+K
Toolbox Icon:	

This command moves the cursor to the midpoint of the closest line or line segment and sets a point there.

Using the Command

Select the **MIDPOINT** command, move the cursor near the line you want to snap to, and click the left mouse button.

Example: Set a point on the midpoint of a line.

Choose the **MIDPOINT** command. Move the cursor near the line in your drawing and click the left mouse button. The cursor snaps to the midpoint of the line and sets a point there.

Note: In 2-D Mode, this command will snap to the midpoint of a line's *projection* on the XY plane, not to the midpoint of the actual 3-D line.

Mirror Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	MIRROR
Point 1:	Location of mirrored object

The Mirror command is used to make a mirror image of a selection.

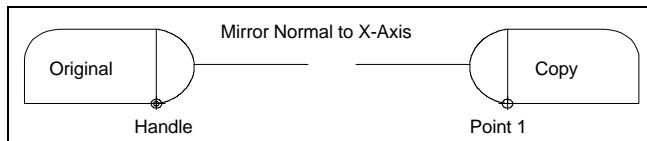
Using the Command

Choose the **MIRROR** command. The dialog box contains five options for the Mirror command:

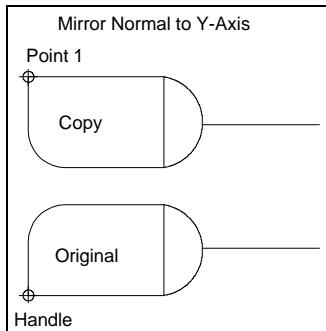


Normal to X Axis

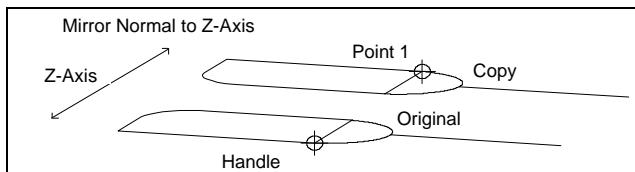
Selecting this option places a mirror which faces in the direction of the X-axis. That is, the mirror lies on the X axis and is perpendicular to it.

**Normal to Y Axis**

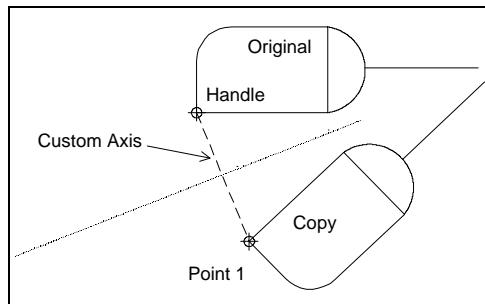
This places the mirror perpendicular to the Y-axis.

**Normal to Z Axis**

This places the mirror perpendicular to the Z-axis.

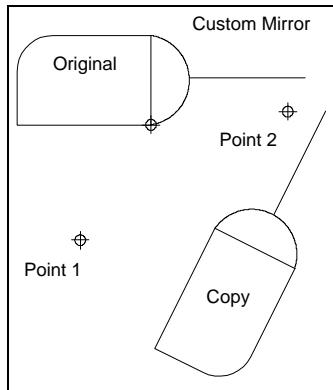
**Custom Axis**

This option specifies a 3-D line perpendicular to the face of the mirror. You can use the option to create custom mirror effects, such as producing a mirror image facing at an angle 90 degrees away from the original.



Custom Mirror

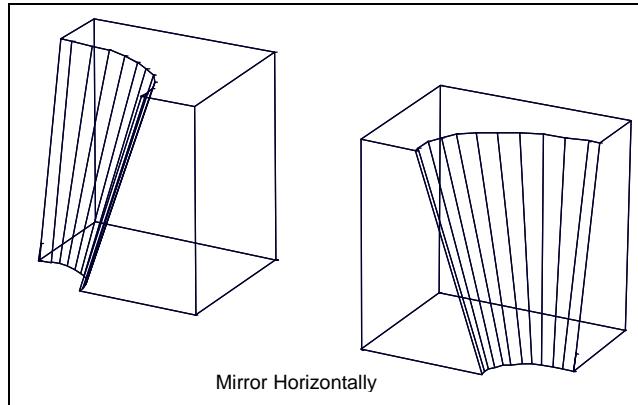
This option mirrors the object based on a mirror defined by two points.



To use the command, set a point for the location of the mirror image. The mirror image will be located at that point, and mirrored according to the option selected. If CUSTOM AXIS is selected, then the mirror will be normal (perpendicular) to the line between the selection handle and the point that was set.

Example: Make a horizontal mirror image of an object.

Select the object and choose the MIRROR command. Select NORMAL TO X AXIS and set a point for the mirrored object's location. The object is mirrored horizontally.

**Most Recently Used Files Command**

Menu:	FILE
Menu Command:	<i>FILENAME</i>

DesignCAD provides you with a list of recently used drawing files. The filenames appear on the list in order of their most recent use. To open one of these files, select it from the menu.

Using the Command

Select the *filename* of a file you have worked on recently from the listing in the **FILE** menu. DesignCAD will open the file whether it was opened during the current drawing session or a previous one. This is a faster way of opening a recently used file than using the Open command and being forced to specify the path to the directory in which it is stored.

Move Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	MOVE
Shortcut Key:	M
Point 1:	New location of the handle
Point 2:	New location of second handle (optional)
Point 3:	New location of third handle (optional)

The Move command moves a selected entity or group to another location. One, two, or three points may be used to specify the new location.

Using the Command

To use the command, select the object or group to be moved. Choose the **MOVE** command. A rubber-band box appears. If the selection has only one handle, the first point you set places the object you are moving at its new location with that handle. The size and orientation remain the same.

If you have placed two or three selection handles with the Set Handles command, you can scale and determine the orientation of the selection with those points. If a second point is used, you can adjust the selection's size and orientation with the second handle. The third point is used to adjust the object's orientation.

Note: In order to use the second selection handle to change the size of the selection, you must uncheck the **Use Original Scale** checkbox.

You can also activate this command by moving the cursor over the primary handle. When the cursor changes into a four-sided arrow, you can move the object in one of two ways:

1. Press the left mouse button and hold. Drag the selection to the new location.
2. Click the left mouse button once to lock the Move command. Drag, or use any of the point commands, to select a new location. If more than one handle has been set for the selection, you can set more than one point when you move the object.

This command is like the Duplicate command except that the selection is moved instead of copied.

Example: Move an object to a new location in the drawing.

Select an object in the drawing. Next, choose the **MOVE** command. Move the cursor to the new location for the object. Set a point for the insertion. If you have set more than one handle for the object, you can set a point for each handle.

Move After Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	MOVE AFTER
Point 1:	Entity to send the selected entity behind

The Move After command sends the selected object behind the object specified. This command lets you change the display order of entities so that the correct portions of overlapping entities are displayed and printed. This command affects entities in 2-D Mode and entities that lie on the same X-Y plane in 3-D Mode.

Using the Command

Select the entity to be sent back in the stack of entities on the screen. Choose the MOVE AFTER command from the SELECTION EDIT submenu in the EDIT menu. Set a point on the entity you would like the selected entity sent behind. The selected entity is positioned behind the entity specified.

Example: Send a filled circle behind a filled box in a stack.

Select a filled circle in a stack of entities to send behind a filled box that lies several back on the screen. Choose the MOVE AFTER command from the SELECTION EDIT submenu in the EDIT menu. Set a point on the filled box. The filled circle is sent directly behind the filled box.

See Also: *Move in Front of Command, Move to Back Command, Move to Top Command*

Move in Front of Command

Menu:	FILE
Submenu:	SELECTION EDIT
Menu Command:	MOVE IN FRONT OF
Point 1:	Entity in front of which to bring the selected entity

The Move in Front of command sends the selected object in front of the object specified. This command lets you change the display order of entities so that the correct portions of overlapping entities are displayed and printed. This command affects entities in 2-D Mode and entities that lie on the same X-Y plane in 3-D Mode.

Using the Command

Select the entity to be sent forward in the stack of entities on the screen. Choose the MOVE IN FRONT OF command from the SELECTION EDIT submenu in the EDIT menu. Set a point on the entity in front of which you would like the selected entity sent. The selected entity is positioned in front of the entity specified.

Example: Send a filled circle in front of a filled box.

Select a filled circle in a stack of entities to send in front of a filled box that lies several forward on the screen. Choose the MOVE IN FRONT OF command from the SELECTION EDIT submenu in the EDIT menu. Set a point on the filled box. The filled circle is sent directly in front of the filled box.

See Also: *Move After Command, Move to Back Command, Move to Top Command*

Move to Back Command

Menu:	FILE
Submenu:	SELECTION EDIT
Menu Command:	MOVE TO BACK

The Move to Back command sends the selected object or objects behind other objects on the screen. This command lets you change the display order of entities so that the correct portions of overlapping entities are displayed and printed. This command affects entities in 2-D Mode and entities that lie on the same X-Y plane in 3-D Mode.

Using the Command

Select the entity to be sent behind others on the screen. Choose the MOVE TO BACK command from the SELECTION EDIT submenu in the EDIT menu. The selected entity is sent behind all of the other entities on the screen.

Example: Send a filled circle behind a filled box.

Select a filled circle that lies in front of a filled box on the screen. Choose the MOVE TO BACK command from the SELECTION EDIT submenu in the EDIT menu. The filled circle sent behind the filled box.

See Also: *Move After Command, Move in Front of Command, Move to Top Command*

Move to Top Command

Menu:	FILE
Submenu:	SELECTION EDIT
Menu Command:	MOVE TO TOP

The Move to Top command brings the selected object or objects in front of the other objects on the screen. This command lets you change the display order of entities so that the correct portions of overlapping entities are displayed and printed. This command affects entities in 2-D Mode and entities that lie on the same X-Y plane in 3-D Mode.

Using the Command

Select the entity to be brought in front of others on the screen. Choose the MOVE TO TOP command from the SELECTION EDIT submenu in the EDIT menu. The selected entity is brought to the very front of the entities on the screen.

Example: Bring a filled circle in front of a filled box.

Select a filled circle that lies behind a filled box on the screen. Choose the MOVE TO TOP command from the SELECTION EDIT submenu in the EDIT menu. The filled circle is brought in front of the filled box.

See Also: *Move After Command, Move in Front of Command, Move to Back Command*

New Command

Menu:	FILE
Menu Command:	NEW
Shortcut Key:	Ctrl+N
Toolbox Icon:	

The New command opens a new drawing document. It does not close any drawing that you already have open. It simply opens an empty window of full-screen size, in which to build the new drawing.

Using the Command

Choose the NEW command. A blank drawing appears on the screen, but it does not close the current drawing. You can switch from one drawing to the next using **Ctrl-F6**.

If you already have several documents open and try to start a new one, DesignCAD 3D MAX may warn you that it's not possible to create an empty document.

In this case, close one or more of your drawings, or other open applications, and try NEW again. The number of documents that you can open depends on the amount of memory on your system, the number of other applications that are open, the complexity of each drawing, the number of view windows you have opened for each drawing, and other factors.

See Also: [Open Command](#), [Close Command](#), [Load Symbol Command](#), [Save Command](#), [Save As Command](#)

New Window Command

Menu:	WINDOW
Menu Command:	NEW WINDOW

The New Window command opens a new view of a drawing. DesignCAD 3D MAX allows several different windows, or views, to be opened simultaneously. These can be zoomed or panned independently of each other.

Using the Command

Choose the NEW WINDOW command from the WINDOW menu. A new drawing window automatically opens. Change the window's view, size, and location as desired.

Open Command

Menu:	FILE
Menu Command:	OPEN
Shortcut Key:	Ctrl+O
Toolbox Icon:	

The Open command opens a drawing file and loads it onto the screen as the current drawing.

Using the Command

Choose the OPEN command. The OPEN dialog box appears. In the FILE NAME box enter or select the name of the file you want to open. In the LOOK IN box select the location of the file. In the FILES OF TYPE box select the type of file you want to open. Click the OK button when you have entered the necessary information, or click the CANCEL button to return to the current drawing.

If you already have a drawing on the screen, it will not close that drawing but will open a second drawing.

If you already have several documents open and try to start a new one, DesignCAD 3D MAX may warn you that it's not possible to another document.

In this case, close one or more of your drawings, or other open applications, and try OPEN again. The number of documents that you can have open depends on the amount of memory on your system, the number of other applications that are open, the complexity of each drawing, the number of view windows you have opened for each drawing, and other factors.

The Ignore View Data option tells DesignCAD to ignore view angles and distances at which the drawing was previously saved. DesignCAD loads the drawing using the default views and viewing angles for the appropriate drawing mode (i.e., 2-D Mode or 3-D Mode).

See Also: **Close Command, Load Symbol Command, New Command, Save Command, Save As Command**

Options Command

Menu:	OPTIONS
Menu Command:	OPTIONS
Shortcut Key:	Q

The Options command gives you the opportunity of setting a wide range of parameters that control the operation of DesignCAD. The Options command displays a number of options folders. For more information, refer to the individual entries listed in the "Command Reference" section of this manual:

- Color Options
- Cursor Options
- Dimension Options
- File Locations Options
- General Options
- Grid Options
- Keyboard Options
- Layer Options
- Light Source Options
- Material Options
- Menu Options

- Text Options
- Toolbox Options
- View Options

Origin Command

Menu:	POINT
Menu Command:	ORIGIN
Point 1:	New location for the drawing's origin

The Origin command can be used to set the origin, (location 0, 0, 0) anywhere in the drawing. This can be convenient if you want to use coordinates relative to a particular point on the drawing.

Using the Command

Choose the ORIGIN command. Set a point anywhere on the screen. The location of this point becomes the new origin. The origin remains the same until it is changed again.

Example: Reset the origin of your drawing.

Select the ORIGIN command. Move the cursor to the lower-left corner of the screen and set a point. The origin of the drawing will move to that location. Notice that the X,Y,Z location on the coordinate bar now reads 0,0,0 at the new origin.

Original Size Command

Menu:	VIEW
Menu Command:	ORIGINAL SIZE
Toolbox Icon:	

The Original Size command lets you restore a zoomed drawing back to its original size.

Using the Command

Choose the ORIGINAL SIZE command from the VIEW menu. The drawing will return back to its original size. This works for drawings that have been zoomed in or zoomed out.

Ortho Command

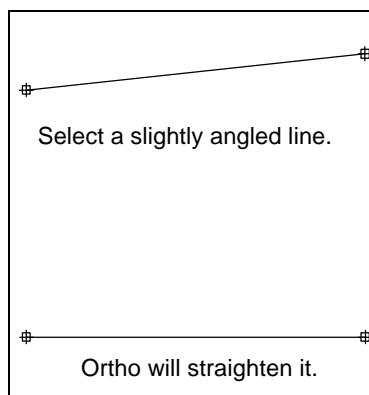
Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	ORTHO
Toolbox Icon:	

The Ortho command forces selected lines to be vertical or horizontal. This command forces all line segments within 10 degrees of parallel with the X, Y, or Z axis to lie exactly parallel to that axis.

Using the Command

Select the line to be straightened. Choose the ORTHO command from the SELECTION EDIT submenu in the EDIT menu. The line will be converted to a vertical or horizontal entity.

Example: Make an angled line horizontal.



Orthogonal Mode Command

Menu:	OPTIONS
Menu Command:	ORTHOGONAL MODE
Shortcut Key:	H
Toolbox Icon:	

Orthogonal Mode constrains consecutive points in drawing commands so that the alignment of a pair of points is parallel to the X, Y, or Z axis. If you set points that are not aligned parallel to one of these axes in Orthogonal Mode, DesignCAD automatically shifts the second point of the pair so that the points are aligned parallel to the nearest matching axis.

Using the Command

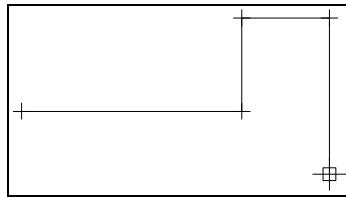
Activate ORTHOGONAL MODE from the OPTIONS menu. Select the LINE command and set a beginning point. Set any number of points to draw the line. Then press **Enter** or double-click the mouse to end the command.

Note: The **Shift** key may be used in conjunction with the right mouse button in Orthogonal Mode while drawing lines to line up the line segments that you are working on.

Example: Draw a line with segments parallel to the X- and Y-axes.

Press **H** to activate the ORTHOGONAL MODE command. Select the LINE command and set a point on the screen. Move the cursor to the right. Notice that the rubber-band line only moves parallel to the X axis. Set a point for the body of the line and move the cursor up. Again the line is drawn exactly parallel to an axis. Move to the right again and set a point. Finally move down below the

first and second points. Press and hold the **Shift** key while pressing the right mouse button. The point is set even with the first and second points. Press **Enter** to complete the command and add the line to the drawing.



Ortho Line Command

Menu: DRAW
 Submenu: LINES
 Menu Command: ORTHO LINE
 Toolbox Icon: 
 Point 1: Beginning of line
 Point 2: End of first segment
 Point 3-n: End of second and subsequent segments (optional)

The Ortho Line command draws lines with segments that are parallel to the X, Y, or Z axis. If you set points that are not parallel to one of these axes, the Ortho Line command automatically shifts the line to be parallel to the nearest matching axis.

Using the Command

Choose the ORTHO LINE command. Set a beginning point for the ortho line. Set any number of points to draw the line. Then press **Enter** or double-click the mouse to end the command.

Example: Draw a line parallel to the X axis.

Select the ORTHO LINE command and set a point on the screen. Move the cursor to the right and up. Notice that the rubber-band line only moves along the X axis. Set a point for the body of the line and move the cursor up. Again the line is drawn exactly parallel to an axis. Set another point and press **Enter** to complete the command and add the line to the drawing.

Ortho Line-2 Command

Menu: DRAW
 Submenu: LINES
 Menu Command: ORTHO LINE-2
 Point 1: Beginning of the line
 Point 2: Direction of 1st segment
 Point 3: Length of 1st segment; length & direction of 2nd segment
 Points 4-n: even number: direction of segment; odd number: previous segment length; current segment length & direction

Like the Ortho Line command, the Ortho Line-2 command draws lines that are parallel to the X, Y, or Z axis. However, the points set in the Ortho Line-2 command function differently.

With the Ortho Line-2 command, the first point set is the beginning of the line. The second point determines only the direction of the first line segment, not the length. After the second point is set, the length of the first line segment continues to change when the cursor moves. The third point determines the direction of the second line segment and the length of both the first and second line segments. After the third point is set, all even numbered points function like the second point, determining only the direction of the line segment. All the following odd numbered points work like the third point and determine the length of the previous line segment and the length and direction of the current line segment.

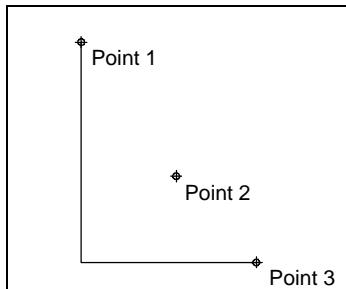
Using the Command

Choose the ORTHO LINE-2 command. Set a beginning point for the line. Move the cursor in the desired direction for the first segment and set a second point. Notice how the rubber-band line is bent in the direction you specified, but the length of both segments continues to change when you move the cursor. When you have both segments the length you want them and the second segment the desired direction, set a third point. Set any number of points to draw the line. Then press **Enter** or double-click the mouse to end the command.

Note: The Ortho Line-2 command only works in 2-D Mode.

Example: Draw a bent line.

Choose the ORTHO LINE-2 command. Set a point for the beginning of the line. Now, move the cursor down and to the right. Notice how the rubber-band line will only move straight along the X-Axis or the Y-Axis. Set a point so that the line goes down along the Y-Axis. Now, move the cursor up and down. Notice how the length of the first segment gets longer or shorter with the movement of the cursor. Move the cursor to the right and set a point so that the line forms an "L" shape. Press **Enter** to complete the command and add the line to the drawing.



Page Setup Command

Menu: FILE
Menu Command: PAGE SETUP

The Page Setup command is used to change some of the DesignCAD printer options. This dialog box can also be accessed by clicking the SETUP button in the Print dialog box.

Paper

Size

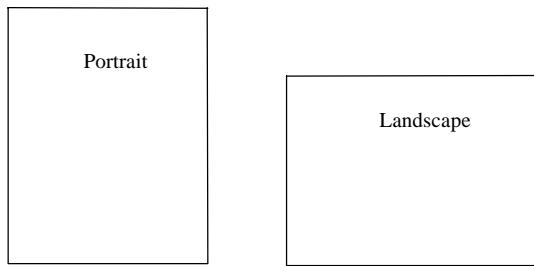
This option allows you to select the paper size for your drawing from a list of standard sizes.

Source

This option allows you to select the paper source from a list of standard printer paper sources.

Orientation

This option determines the orientation of the drawing. A drawing can be printed using Portrait or Landscape orientation.



Margins

Top

This option is used to set the size of the top margin.

Bottom

This option is used to set the size of the bottom margin.

Left

This option is used to set the size of the left margin.

Right

This option is used to set the size of the right margin.

Printer

Clicking on this button brings up the dialog box for selecting the printer for the pending print job. Select the printer by clicking on the down arrow to the right of the PRINTER: label and clicking on the printer name. If you do not see the printer you would like to use for the print job in the list of available printers, make sure the printer is installed correctly with your version of Windows. If you have problems, consult your Windows documentation and your printer manual. Make sure you have not only the correct driver installed, but also that it is the most recent driver from the manufacturer.

OK

Clicking on this button saves the changes you have made and returns you to your drawing.

Cancel

Clicking on this button disregards the changes you have made and returns you to your drawing.

Pan Command

Menu:	VIEW
Menu Command:	PAN
Shortcut Key:	Ctrl+Shift+P
Toolbox Icon:	
Point 1:	Specific point to be moved
Point 2:	New screen location for Point 1

The Pan command is used to slide the drawing around on the screen. To pan a drawing, select the command and then drag the cursor across the screen. The drawing will be dragged to its new location.

Using the Command

Choose the Pan command. The cursor turns into a four-headed arrow. Drag the mouse across the screen until the drawing is in the position you want. Then release the mouse button.

You can also set a point for the "source" and "destination" of the drawing. The first point is the original location and the second point is the new location for that part of the drawing.

Example: Slide your entire drawing around on the screen.

Select the PAN command and set a point on one of the objects. Move the cursor around the screen and notice how all objects in the drawing move as a group. When the drawing is positioned to your liking, set the second point.

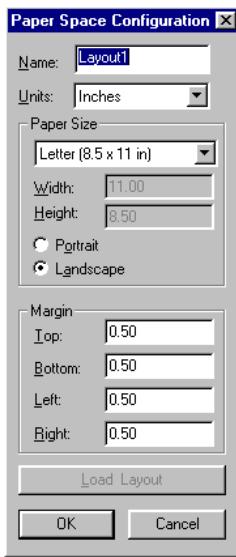
Paper Space Configuration Command

Menu:	FILE
Submenu:	PAPER SPACE VIEW FRAME SETUP
Menu Command:	PAPER SPACE CONFIGURATION

The Paper Space Configuration command sets up the area to be used for the Paper Space Mode. This command should be used to ensure that the Paper Space that is about to be created will print correctly when the time comes.

Using the Command

Choose the PAPER SPACE CONFIGURATION command from the PAPER SPACE VIEW FRAME SETUP submenu in the FILE menu.



Name

This is the name of the currently selected Paper Space Template.

Units

This option changes the unit of measurement used for specifying the margins for the Paper Space Configuration. Notice that the HEIGHT and WIDTH values also automatically convert to the selected unit of measurement.

Paper Size

Use the list box at the top of the Paper Size options area to specify the paper size that your printer uses.

Width

Displays the width of the paper size selected in the units of measurement that are currently selected in the Units list box.

Height

Displays the height of the paper selected in the units of measurement that are currently selected in the Units list box.

Portrait and Landscape

These options determine how you want your printer to use the paper when the paper space is printed. The paper space can be setup to use Portrait or Landscape orientation. These options also affect the paper boundaries in the Paper Space Template.

Margin

Top

The size of the top margin for the Paper Space Configuration.

Bottom

The size of the bottom margin for the Paper Space Configuration.

Left

The size of the left margin for the Paper Space Configuration.

Right

The size of the right margin for the Paper Space Configuration.

Load Layout

Opens the Paper Space Template dialog box which allows the selection of a previously saved Paper Space Template. When all of the options have been set for the Paper Space Configuration and the **OK** button is pressed, the template is enlarged or reduced to the maximum scale that will fit on the paper size selected for the Paper Space Configuration.

OK

Clicking on this button saves the changes you have made and opens Paper Space Mode.

Cancel

Clicking on this button disregards the changes you have made and returns you to your drawing.

Paper Space Mode Command

Menu:	FILE
Submenu:	PAPER SPACE VIEW FRAME SETUP
Menu Command:	PAPER SPACE MODE
Toolbox Icon:	

The Paper Space Mode command makes it possible to print multiple views of the same drawing in a single print job. Options such as perspective, shading, light source, focal point, view settings, and zoom can be set for each separate view created in the Paper Space.

Using the Command

Choose the PAPER SPACE MODE command from the PAPER SPACE VIEW FRAME SETUP submenu in the FILE menu. This will display the drawing in the currently selected Paper Space Template. If no template has been selected, the Paper Space Template dialog box appears. Select the template you wish to use and click **OK**. Clicking **OK** without selecting one of the existing templates will give you a blank template. A blank template will display nothing until you create one or more customized view frames using the CREATE VIEW FRAME command. The Create View Frame command can be accessed from the Paper Space dialog box or the View Frame menu which appears once Paper Space Mode is enabled.

There are several options available in the dialog box which can be set for each view frame.





Create View Frame

Used to create a printable view window for which you set the placement, size, and view options.



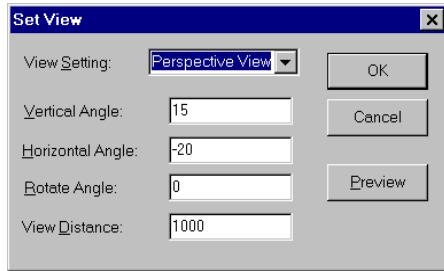
Pick an Entity as View Frame

Uses simple 2-D entities that have been drawn while in Paper Space Mode as view frames. Just draw the entity, click on this option and then click on the entity you just drew. That entity is used as a view frame for the drawing.



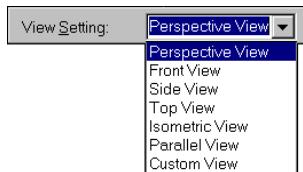
Change View Setting

Several different view options for an individual View Frame can be set using this command.



View Setting

Determines which view of the drawing is shown in the View Frame. The views listed are the same as those in the Projection list-box in the View toolbox.



Vertical Angle

Sets the vertical angle of the drawing.

Horizontal Angle

Sets the horizontal angle of the drawing.

Rotate Angle

Rotates the angle of the drawing.

View Distance

Sets the view distance.

Preview

This command is used to preview the view setting changes made to the current View Frame of the drawing.

**Draw Outline Box**

Displays or hides an outline border around the selected View Frame. This command works as a toggle, showing or hiding the border.

**Fit to View Frame**

This command works like the Fit to Window Command, centering the drawing in the View Frame with all objects visible.

**Center on View Frame**

To center the drawing in the View Frame without changing the zoom factor, use the this command instead of the Fit to View Frame command.

**Fit Drawing to Paper**

This resizes the view frames so that all view frames will fit within the paper space at the largest possible size.

**Set Focal Point**

Drags the drawing in the View Frame by its center point.

**Pan Drawing**

Slides the drawing across the View Frame using a reference point and the new location for that point.

**Light Source & Layer Settings**

Opens the Light Source and Layer Options folders so that these options may be set for the current View Frame. For details on the Light Source and Layer Options folders, see their command entries in the "Command Reference" section of this manual. Changes made to these options will only affect the View Frame, not the actual drawing.

**Section Zoom**

This command uses a zoom window to zoom in on a specific area of the drawing.

**Zoom In**

Automatically zooms in at increments preset by the program.



Zoom Out

Automatically zooms out at increments preset by the program.



Change Zoom Factor

Changes the zoom factor of the currently selected view frame interactively. Click on this option. Move the cursor into the Paper Space area. Press and hold the mouse button. Move the cursor up to increase the zoom factor or down to decrease the zoom factor. When the desired appearance is achieved, release the mouse button.



Save Current Layout as Template

Saves the current Paper Space Template, including all View Frames and the options set for each.



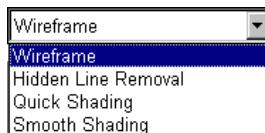
Preview

This option shows a preview of how the printed Paper Space will look including shading and hidden line removal.

Note: To see the page layout including margins, use the Print Preview command on the Toolbar.

Display Method

This option is used to designate a display method for the current view frame. The changes to the Display Method will only be seen with the Preview command in the Paper Space dialog box.



Paper Space Layout

This list box lets you select a different Paper Space template in which to display the drawing.



Edit Layout

This button opens the Paper Space Template dialog box so you can change the paper size and margins for the current template.



Add a New Layout

Brings up the Paper Space Template dialog box so a template can be added to the Paper Space Layout list box.



Delete Current Layout

Deletes the currently selected template in the Paper Space Layout list box.

See Also: *Automatic Rendering Command*

Parallel Command

Menu: DRAW
 Submenu: LINES
 Menu Command: PARALLEL
 Shortcut Key: =
 Toolbox Icon:

Point 1: Existing line
 Point 2: Location for parallel line

The Parallel command draws a line parallel to any existing line, curve, or arc.

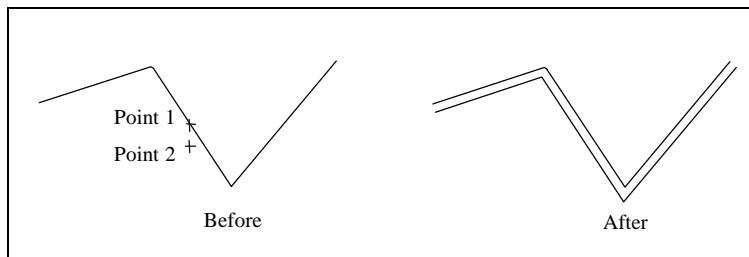
Using the Command

Choose the PARALLEL command. Set a point on the original line and a point for the location of the parallel line.

This command works on three-dimensional lines as well as lines that lie on a single plane.

Example: Draw a line parallel to a multi-segment line.

Select the PARALLEL command. Set a point on the line and move the cursor away from it. A rubber-band line is drawn parallel to the first line at the current cursor distance. When the line is where you want it, set the second point.



Parallel by Distance Command

Menu: DRAW
 Submenu: LINES
 Menu Command: PARALLEL BY DISTANCE
 Toolbox Icon:

- Point 1: Line to be paralleled.
- Point 2: Direction of parallel line from original line.
- Point 3: Location of next parallel line (optional)

The Parallel by Distance command draws a parallel line a specified distance from any other line or curve. The distance of the parallel is set in the dialog box. The direction of the parallel from the original line is established by setting a point on the screen.

Using the Command

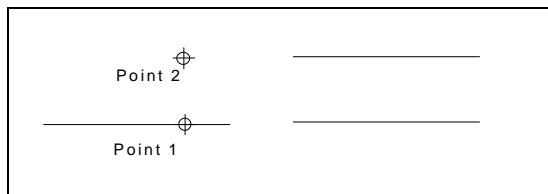
Choose the PARALLEL BY DISTANCE command in the Toolbox. Enter how far you want the parallel from the original line in the DISTANCE box in the dialog box.



Set a point on the original line. A rubber-band line shows how the parallel line will look. Next, move your cursor to the side of the original line on which you want the parallel to be drawn and set a point. A parallel line will be drawn at the specified distance. Press **Enter** to end the command.

Example: Draw a parallel line 4 units from another line.

Choose the PARALLEL BY DISTANCE command in the Toolbox. Enter 4 in the DISTANCE box in the dialog box. Set a point on a line in your drawing. Move the cursor to the side of the line on which you want the parallel to be drawn. A rubber-band line shows how the parallel line will be drawn. Click the left mouse button to draw the parallel line and press **Enter** to finish.



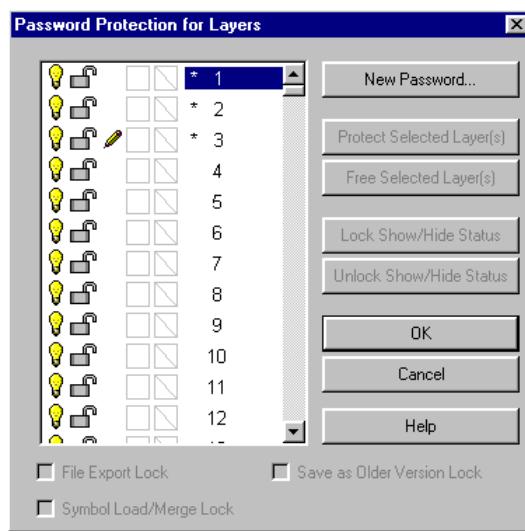
Password Protection Manager Command

Menu:	OPTIONS
Menu Command:	PASSWORD PROTECTION MANAGER

The Password Protection Manager lets you set up a password for a drawing. Once a password has been established, you can lock the editability and visibility of specific layers of the drawing. You can also prevent the file from being exported as another file type, loaded as a symbol, or saved as an older DesignCAD file.

Using the Command

Open the file you want to password protect. Select the PASSWORD PROTECTION MANAGER command. The Password Protection Manager for Layers dialog box appears.



Click the NEW PASSWORD button. The Password Change dialog box appears.



Enter the new password in the NEW PASSWORD text box and again in the CONFIRM NEW PASSWORD text box. Passwords can be up to 50 characters in length and are case sensitive. Click the OK button. DesignCAD returns to the Password Protection for Layers dialog box.

Warning: Do not lose your password. There is no way of recovering a password-locked drawing without the password.

Once the password has been established, the Protect Selected Layer(s), Free Selected Layer(s), Lock Show/Hide Status, and Unlock Show/Hide Status buttons may be used to protect selected layers in the list on the left side of the dialog box. To select multiple layers, click on a layer number; then press and hold the **Ctrl** key on the keyboard while clicking on additional layer numbers.

File Export Lock

If the File Export Lock is enabled, DesignCAD will not export a copy of the drawing unless the password has been given in the Password Protection Manager.

Symbol Load/Merge Lock

If the Symbol Load/Merge Lock is enabled, DesignCAD will not load a copy of the drawing into another drawing as a symbol. To disable the lock, open the original drawing, run the

PASSWORD PROTECTION MANAGER command, give the password, disable the SYMBOL LOAD/MERGE LOCK option, click the OK button, and save the drawing.

Save as Older Version Lock

If the Save as Older Version Lock is enabled, DesignCAD will not save a copy of the drawing as an older DesignCAD drawing version unless the password has been given in the Password Protection Manager.

To disable password protection, click the NEW PASSWORD button. Leave the NEW PASSWORD and CONFIRM NEW PASSWORD text boxes blank and click the OK button. A dialog box appears to inform you that all password protection settings will be removed and ask if you are sure that you want to remove password protection. Click the YES button.

See Also: *Layer Options*

Paste Command

Menu: EDIT
 Menu Command: PASTE
 Shortcut Key: **Ctrl+V**
 Toolbox Icon: 

Point 1: First handle (lower-left-front corner)
 Point 2: Second handle (lower-right-front corner)
 Point 3: Third handle (lower-right-rear corner)

The Paste command copies drawing objects from the Windows Clipboard into DesignCAD 3D MAX.

Using the Command

Choose the PASTE command. Set a point for the location of the object or group to be pasted into the drawing. If necessary, a second point can be used to specify the angle and size of the object, and a third point can be used to set the 3-D orientation.

Note: In order to use the second selection handle to change the size of the selection, you must uncheck the Use Original Scale checkbox in the Command Line.

Example: Suppose that you want to insert an image in your drawing that you had put in Clipboard.

Select the PASTE command. A rubber-band box representing the scale of the object will follow the cursor as you move it around the screen. When you have selected a location for the copy of the object, set a point. Move the cursor to the right to stretch the scaling box. Set points for the lower-right-front and lower-right-rear handles of the copy, or press **Enter** to insert the drawing at the original scale.

See Also: *Cut Command, Copy Command, Import Command*

Pause Recording Command

Menu:	TOOLS
Menu Command:	PAUSE RECORDING

The Pause Recording command suspends the recording of a macro. Click the PAUSE button if you want to pause during the recording of a macro.

Using the Command

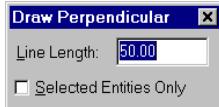
While recording a macro, choose the PAUSE RECORDING command. The macro stops recording until you choose the CONTINUE RECORDING command.

See Also: **Macro Record Command, Stop Recording Command, Continue Recording Command**

Perpendicular From a Line Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	PERPENDICULAR FROM A LINE
Toolbox Icon:	
Point 1:	Reference point
Point 2:	Direction of perpendicular

This command draws a line perpendicular to an existing line. The perpendicular line is drawn from a point on or near the existing line in the direction of the second point set. You can enter the length for the perpendicular line in the dialog box.



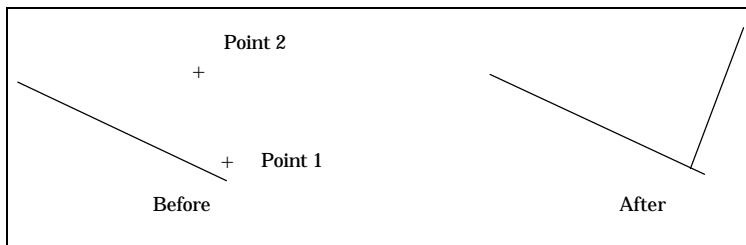
Using the Command

Choose the PERPENDICULAR FROM A LINE command. Set a reference point on or near the line from which you want the perpendicular line drawn. A rubber-band line appears. Then set a second point away from the first line. A perpendicular line is drawn from the existing line through the first point in the direction of the second point.

This command is similar to the Perpendicular to a Line command, but the Perpendicular From a Line command uses the point on or near the existing line for the perpendicular location. The Perpendicular to a Line command uses the point away from the line as the reference for the perpendicular location.

Example: Draw a 20-unit line perpendicular from a certain point on another line in your drawing.

Select the PERPENDICULAR FROM A LINE command. Enter **20** for the LENGTH in the dialog box. Set a point on or near the line, at the point through which you want the perpendicular line drawn. A rubber-band line, 20 Drawing Units in length, will be drawn from the line and through Point 1. If you move the cursor to the other side of the line, the perpendicular line will flip to the other side as well. Set the second point when the line extends in the desired direction.



Perpendicular Plane Command

Menu:	DRAW
Submenu:	PLANES
Menu Command:	PERPENDICULAR PLANE
Toolbox Icon:	
Point 1:	Existing line
Point 2:	Point on the line to form the center of the plane

The Perpendicular Plane command can be used to quickly create planes perpendicular to any given point on a line or curve.

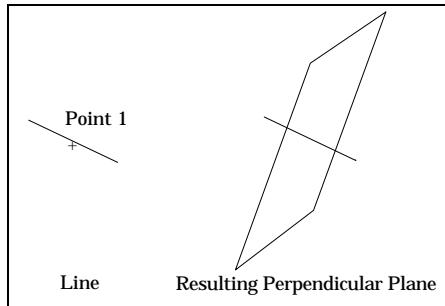
Using the Command

Choose the Perpendicular Plane command. You can set the width of the created plane in the PLANE WIDTH box in the dialog box. Set a point on the existing line. Then set a point on the line to establish the center of the plane.



Example: Draw a 20-unit plane perpendicular to a line.

Select the PERPENDICULAR PLANE command and enter **20** for the PLANE WIDTH. Set a point near the middle of the line. DesignCAD will draw a plane of the specified width perpendicular to the line.



Perpendicular to a Line Command

Menu: DRAW

Submenu: LINES

Menu Command: PERPENDICULAR TO A LINE



Toolbox Icon:

Point 1: End point of perpendicular line

Point 2: Point on line to which new line is drawn perpendicular

The Perpendicular to a Line command draws a line perpendicular from a point to an existing line.

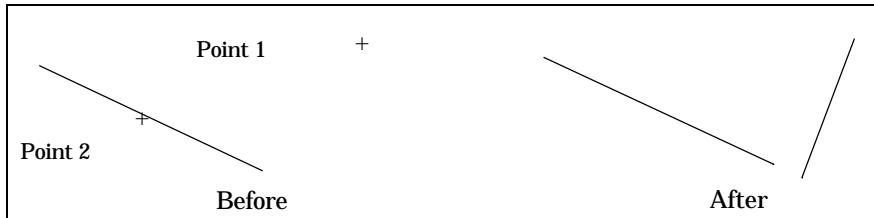
Using the Command

Choose the Perpendicular to a Line command. Set one point for the location and starting point of the perpendicular line and a second point on the line to which the perpendicular line will be drawn.

This command is similar to the Perpendicular From a Line command, but the Perpendicular to a Line command uses the point away from the line for the perpendicular location. The Perpendicular From a Line command, however, uses the point on the existing line for the perpendicular location.

Example: Draw a line from a specific point, perpendicular to a line.

Select the PERPENDICULAR TO A LINE command. Now choose a location for the endpoint of the perpendicular line and set a point there. Move the cursor to the line, and a rubber-band line will be drawn from Point 1 perpendicular to the line. Set the second point to insert the perpendicular line.



Plane Command

Menu: DRAW
 Submenu: PLANES
 Menu Command: PLANE
 Shortcut Key: P

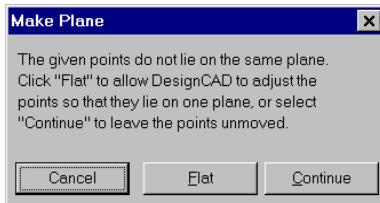


Toolbox Icon:
 Points 1-2: Starting point and edge of the plane
 Points 3-n: Points around the perimeter of the plane

The Plane command is used to draw a plane or flat surface by setting points around the perimeter of the surface. The points of a plane must lie on a single plane in 3-D space. You cannot, for example, have a plane entity with a bump in it. You can use surface commands such as Connect and Extrude for that.

Using the Command

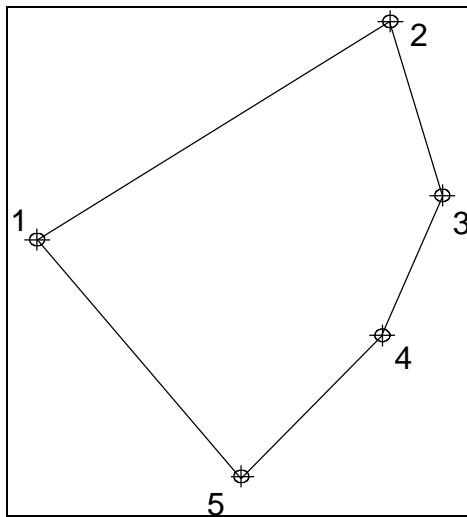
Choose the PLANE command. Set a starting point for the plane. Set another point for one edge of the plane, and then set at least one more unique point (i.e., not on Point 1 or Point 2) to set the plane. When you have set all the points you want, press **Enter** to end the command. DesignCAD checks to see if the points form a valid plane. If they do not, you will have the option to allow DesignCAD to adjust the points so that they form a plane.



Note: Be careful not confuse the Plane command with the Line command. Plane entities have surface area, like a piece of paper, and can be shaded. Line entities have no surface area, like a piece of thin wire, and they cannot be shaded.

Example: Create a plane along four specific points in your drawing.

Select the PLANE command. Set the first point. After the first point is set, a rubber-band drawing of the plane appears, showing how the plane will look when subsequent points are added. Press **Enter** after all the points have been set and a plane is drawn on the screen.



Plane Snap Command

Menu: POINT
 Menu Command: PLANE SNAP
 Shortcut Key: F7
 Toolbox Icon:

Point 1: Point on or near the plane to which you want to snap

The Plane Snap command moves the cursor to the nearest location on the nearest surface and sets a point there.

Using the Command

Choose the PLANE SNAP command and set a point near the plane you want a point on. The cursor snaps to that plane and sets the point there.

Example: Set a point on the nearest point of the closest plane.

Select the PLANE SNAP command. Move the cursor near the plane and click the left mouse button. The cursor snaps to the nearest point on the plane and sets the starting point for your line there.

Plane Subtract Command

Menu: EDIT
 Menu Command: PLANE SUBTRACT
 Point 1: Plane to erase
 Point 2: Plane to be modified

The Plane Subtract command subtracts one plane from another. It can be used to make a hole or opening of a specific shape in a plane.

Using the Command

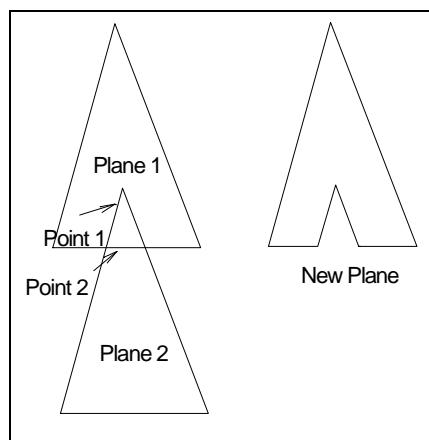
Choose the PLANE SUBTRACT command. Set a point on the plane to be subtracted and a second point on the plane from which the first plane is to be subtracted. The first plane is then subtracted from the second.

If the first plane is in front of or behind the second, then the second plane will be projected onto the first for the subtraction.

Note: If the first plane is completely inside the second plane, then DesignCAD has to cut the second plane into two pieces to perform the subtraction. This is an unavoidable necessity of the algorithm.

Example: Cut a plane with an overlapping plane.

Select the PLANE SUBTRACT command and set a point on the plane you want to subtract. Set a second point on the other plane. DesignCAD removes all of Plane 1 and the portion of Plane 2 that was overlapped.



Point Control Commands

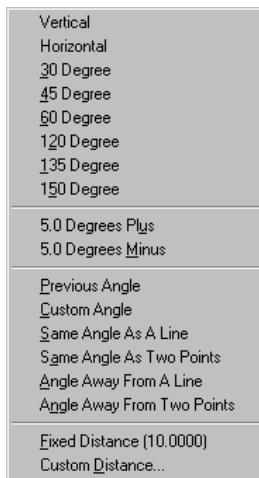
Menu:	POINT
Submenu:	POINT CONTROL
Menu Command:	<i>commandname</i>
Shortcut Key:	J

The Point Control submenu contains several simple commands. These commands can be used after the first point is set in a drawing command to insert another point relative to the first by a specified angle or distance.

Using the Command

After selecting a drawing command and setting the first point, select POINT CONTROL from the POINT menu. A menu of simple commands appears. Choose the desired command from this listing. The movement of the cursor is restricted according to the command.

Note: The Custom Distance command is available in 2-D mode and 3-D mode. The other Point Control commands are available in 2-D mode only.



Vertical

This command forces the cursor to move parallel to the Y axis.

Horizontal

This command forces the cursor to move parallel to the X axis.

30 Degrees

The 30 Degrees command restricts movement of the cursor to a 30-degree angle.

45 Degrees

The 45 Degrees command restricts movement of the cursor to a 45-degree angle.

60 Degrees

The 60 Degrees command restricts movement of the cursor to a 60-degree angle.

120 Degrees

The 120 Degrees command restricts movement of the cursor to a 120-degree angle

135 Degrees

The 135 Degrees command restricts movement of the cursor to a 135-degree angle.

150 Degrees

The 150 Degrees command restricts movement of the cursor to a 150-degree angle.

5.0 Degrees Plus

The 5.0 Degrees Plus command restricts the cursor movement to 5.0 degrees more than the previous angle. Two points must be set with one of the drawing commands before this command is accessible. These two points determine the previous angle to which the 5.0 Degrees More command adds 5.0 degrees.

5.0 Degrees Minus

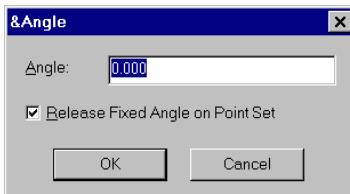
The 5.0 Degrees Minus command restricts cursor movement to 5.0 degrees less than the previous angle. Two points must be set with one of the drawing commands before this command is accessible. These two points determine the previous angle from which the 5.0 Degrees Less command subtracts 5.0 degrees.

Previous Angle

Before becoming accessible, the Previous Angle command requires that **two** points be set with one of the drawing commands. These first two points determine the angle at which the Previous Angle command sets the third point. The third point is set so that a straight line could be drawn through all three points.

Custom Angle

The Custom Angle command restricts cursor movement to an angle entered in the Angle dialog box. Leave the Release Fixed Angle on Point Set option checked to release the cursor when you set a point. Uncheck the checkbox to use the Custom Angle command for several points along the same Custom Angle.



Same Angle as a Line

After setting the first point in a drawing command, choose this command and set a point on an existing line. The cursor is restricted to the angle defined by the line for the insertion of the second point.

Same Angle as Two Points

After choosing the Same Angle as Two Points command, set two points. The Same Angle as Two Points command restricts the cursor to the angle defined by these two points.

Angle Away from a Line

The Angle Away from a Line command lets you place a point on a line. The cursor is restricted to a specified angle away from that line. Set the desired angle in the dialog box.

Angle Away from Two Points

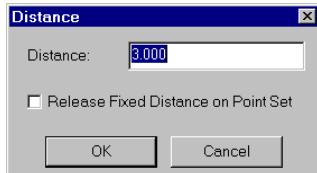
The Angle Away from Two Points command allows you to set two points that define an angle. If an imaginary line were drawn through the two points, the cursor is restricted to a specified angle away from that imaginary line. Set the desired angle in the dialog box.

Fixed Distance

The Fixed Distance command lets you set a point a fixed distance from the previous point. The distance used by the Fixed Distance command may be changed by using the Custom Distance command.

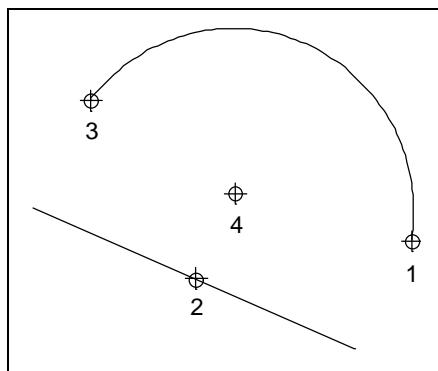
Custom Distance

The Custom Distance command lets you change the distance used in the Fixed Distance command. Uncheck the Release Fixed Distance on Point Set option if you want to set multiple points using the Fixed Distance command.



Example: Set the second point for an arc using the Same Angle as a Line command.

Choose the ARC (ENDPOINTS, CENTER) command. Set the first point for the arc. Choose the SAME ANGLE AS A LINE command from the POINT CONTROL submenu of the POINT menu. Set a point on the existing line that lies at the desired angle. The cursor is restricted to that angle. Set the second point for the arc. Set a third point for the radius of the arc. The angle between the two endpoints of the arc is the same as that of the line.



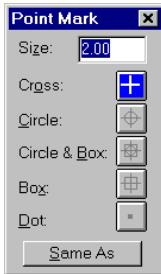
Pointmark Command

Menu:	DRAW
Menu Command:	POINTMARK
Toolbox Icon:	
Point 1:	Location for point mark.

The Pointmark command marks a point in the drawing with a small cross, box, circle or combination of these elements.

Using the Command

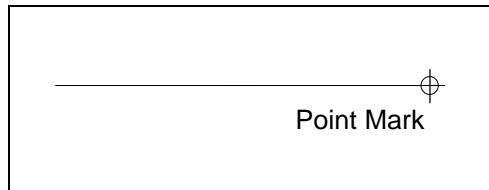
Choose the POINTMARK command in the Toolbox. Select the marker style by clicking a button in the dialog box.



Enter the size in the SIZE box. To draw the point mark the same size as another point mark in the drawing, click the SAME AS button. Then click a point mark in the drawing. Set a point where you want the point mark. A mark is placed there.

Example: Set a point mark on the end of a line.

Draw a line with the Line command. Choose the POINTMARK command Select a point mark style in the dialog box. Move the cursor to the right end of the line and set a point on the endpoint. A mark is inserted at that point.



Point Move Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: POINT MOVE
 Shortcut Key: * (asterisk)

Point 1: Set a point on the point to be moved
 Point 2: Set a point for the new location of that point

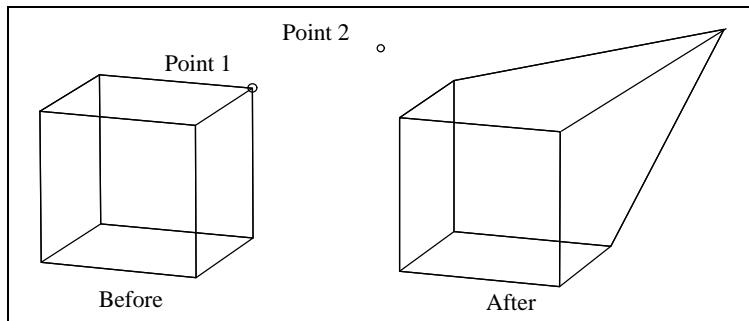
The Point Move command is used to move a point in the drawing.

Using the Command

Set a point directly on another point in the drawing. The Gravity Point command can be used for this. Then set another point at the location to which the first point is to be moved. Every entity in the drawing containing the first point will be redrawn using the second point instead.

Example: Move a corner point of a box.

Select the POINT MOVE command. Move the cursor near one of the corners of the box and click the right mouse button to set a GRAVITY point on it. Now, move the cursor to another location and set a point. The box will be redrawn with that corner in the new location. All other points of the box will remain in place.



See Also: Point Select Mode Command

Point Polar Command

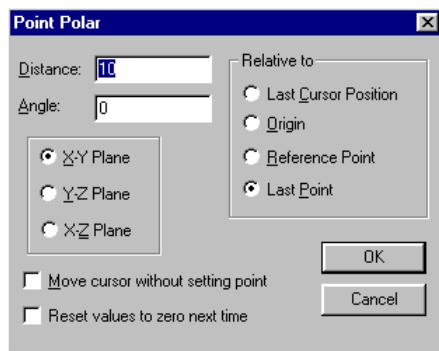
Menu: POINT
 Menu Command: POINT POLAR
 Shortcut Key: ; (semicolon)

This command is used to set a point by specifying the distance and angle (polar coordinates) of the new point from the last point set.

The Point Polar command is used inside another drawing command (such as Line) to set a point at a specific angle and distance from the last point set or the last cursor position.

Using the Command

Choose the Point Polar command. In the Point Polar box enter the DISTANCE and ANGLE from the previous point.



You can specify the angle in either the x-Y (horizontal-vertical), Y-Z, or X-Z plane. You can also specify whether you want the new point to be relative to the LAST CURSOR POSITION, ORIGIN, REFERENCE POINT, or LAST POINT by selecting the options you want.

The options available depend on how you select the command. If you move the cursor to the pull-down menu and select the command, you will not have the option of Last Cursor Position

because the cursor is off the screen. The same occurs if you use the keyboard (Alt+P, then A) to activate the command. If you have set no points and you activate the command from the Command Menu, the point is placed relative to the origin or a reference point. If you press the ; (semicolon) shortcut key, then you have the Last Cursor Position, Origin, and Reference Point options.

You can move the cursor to the new location without setting a point by selecting that option.

If the RESET VALUES TO ZERO NEXT TIME option is selected, the Distance and Angle values will be reset the next time the command is used. If the RESET VALUES TO ZERO NEXT TIME option is not selected, DesignCAD will remember the Distance and Angle values from the current execution of the command and default to those the next time the command is used.

Example: Draw a 20-unit line in the X-Y plane at an angle of 45 degrees.

Select the LINE command and set the first point. Then press the ; (semicolon) key to choose the POINT POLAR command. Enter 20 for DISTANCE and 45 for ANGLE in the edit boxes in the dialog box. Click the X-Y PLANE option and the LAST POINT option. Press **Enter** to execute the command. The second point of your line will be 20 Drawing Units away from the first at a 45-degree angle. Press **Enter** to insert the line into your drawing.

See Also: *General Options—Mathematical Angles, Geographical Angles*

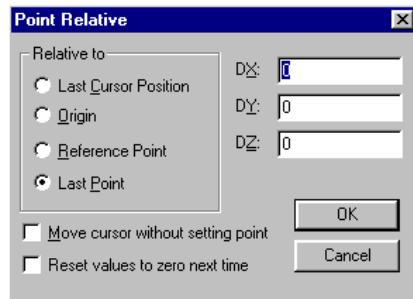
Point Relative Command

Menu:	POINT
Menu Command:	POINT RELATIVE
Shortcut Key:	' (single quote)

The Point Relative command is used to set a point relative to the last point set, the origin, a reference point, or relative to the cursor position. The position of the new point is given in horizontal (X), vertical (Y), and depth (Z) distances from the specified location.

Using the Command

Choose the POINT RELATIVE command. When the Point Relative box appears, enter the X coordinate in the DX box, the Y coordinate in the DY box, and the Z coordinate in the DZ box. Normally you'll want to use the LAST CURSOR POSITION option. Click the OK button when you're satisfied with your choices.



If you move the cursor to the pull-down menu and choose the command, the Last Cursor Position option will not be available because the cursor is off the screen. The same occurs if you use the keyboard (Alt+P, then R) to activate the command. If you have set no points and you activate the command from the Command Menu, the point is placed relative to the origin or a reference point. If you press the ' (apostrophe) shortcut key, then you have the Last Cursor Position, Origin, and Reference Point options.

You can move the cursor to the new location without setting a point by selecting that option.

If the RESET VALUES TO ZERO NEXT TIME option is selected, the DX, DY, and DZ values will be reset to zero the next time the command is used. If the RESET VALUES TO ZERO NEXT TIME option is not selected, DesignCAD will remember the DX, DY, and DZ values from the current execution of the command and default to those the next time the command is used.

Example: Draw a circle with a radius of exactly 10 Drawing Units.

Select the CIRCLE (CENTER, OUTSIDE) command and set a point for the center. Then press the ' (apostrophe) key to select the POINT RELATIVE command. Enter 10 in the DX box. Your circle will have a radius of exactly 10 Drawing Units.

Point Select Mode Command

Menu:	OPTIONS
Menu Command:	POINT SELECT MODE
Shortcut Key:	Ctrl+1
Toolbox Icon:	
Point 1:	Set a point on the entity you want to select.

With Point Select Mode active, you can: select one or more objects; move individual points; select a group of points and move all of them at once (even in more than one object); add or delete points to and from a single selected object; or cut a line or curve into two pieces.

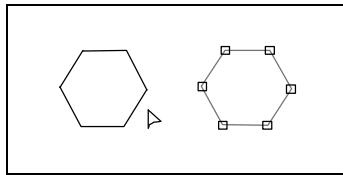
Selecting Objects

To select an individual object in Point Select Mode, simply click on the object. If another item is selected already, the first will be deselected and the second will be selected instead. You can press the **Shift** key and click on other items to add them to the selection set. **Ctrl+Click** excludes points from the selection set. **Ctrl+Shift+Click** toggles selection of points.

Note: Point Select Mode will not allow you to select objects with a selection rectangle. The reason for this is that if multiple points are to be manipulated once the objects are selected, the points are specified with a selection rectangle.

Moving Individual Points in Selected Objects

Select the object. If you are not in Point Select Mode, choose the command from the OPTIONS menu, or press **Ctrl+1**. The points in the selected object are highlighted with small boxes, and the cursor turns into an arrowhead with no tail.



Move the cursor over the point you wish to move. The cursor changes shape into a circle with cross hairs through its center. Click on the point, then move the cursor to the desired position and set a point there. The advantage of this method is that once you have clicked on the point, you can use Point Polar, Point Relative, or Point XY (Point XYZ in 3-D Mode) to move the point to an exact location. Alternatively, you can click on the point and hold down the mouse button and drag the point, only releasing the left mouse button when the point is where you want it.

Moving a Group of Points

Select the object(s) to be manipulated. Press the left mouse button and drag a selection rectangle around the points you wish to select. The small boxes turn solid for each point selected in this manner. If the points do not all lie in a rectangular region, select some with a rectangle, then press **Shift** and drag a selection rectangle around the other points.

Note: You must select even single points using the Shift key and a selection rectangle if you wish to add them to a set of selected points.

Now that your points are all selected, move the cursor over any of the selected points. The cursor turns into a circle with cross hairs. Click once and move the cursor; DesignCAD shows a rubber-band representation of the moving points. Set a second point for your desired destination. All of the selected points are moved, and the shapes of the objects containing those points are updated.

Adding a Point to a Single Selected Entity

Select the entity and enter POINT SELECT MODE. Move the cursor near the object you want to modify. When you are in range the cursor will turn into an arrow with a "+" sign in the arrowhead. Click at the location where you want to add a point. As you move the cursor, a rubber-band line will follow, indicating the shape the object will have. Click again to place the new point where desired. To add the point without changing the shape of the object, set the first point, then use the LINE SNAP command to snap to the same location (Gravity will not work, because the point is not defined yet!).

Note: This function only works for lines or curves and when there is only one item selected. Entities that cannot be modified this way will never show the arrow cursor with the "+".

Extending the End of a Selected Line

Select the line and switch to POINT SELECT MODE. Move the cursor near one endpoint, and press the **Shift** key. If this is the first or last segment of the line, the cursor will turn into an arrow with three dots at its tip. Click the line, and release the Shift key. As you move the cursor, a rubber-band line shows the line extended to the cursor's location. You can shorten or lengthen the segment, or make it go backwards. When the rubber-band line is the desired length, click again. The extended line will be redrawn.

Note: This function only works on lines, and only on the first or last segment.

Deleting a Point from a Selected Object:

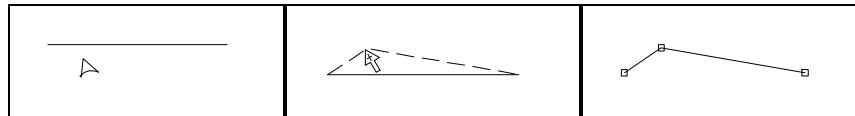
Select the object. If DesignCAD is not in Point Select Mode, choose the POINT SELECT MODE command. Press **Ctrl+Shift**. Move the cursor over the point you wish to delete. The cursor turns into a cursor with a - on it. Click on the point. The point is erased, and the object is redrawn.

Cutting a Selected Object into Two Pieces:

Select the object (this works with a line or curve, also with an arc, or circle if they are saved as a line). If DesignCAD is not in Point Select Mode, choose the POINT SELECT MODE command. Press **Ctrl**. Move the cursor to where you want to cut the object (this does not have to be a pre-defined point). The cursor becomes a tiny pair of scissors. While pressing **Ctrl**, click the left mouse button. The object is cut at the selected point.

Example: Add a point to a line.

If DesignCAD is not already in Point Select Mode, choose the POINT SELECT MODE in the Toolbar. Move the cursor near the line. Click the left mouse button to select the entity. Once the points are showing, move the cursor onto the line. The cursor will have a + in it when it is on the line. Click the left mouse button. Move the cursor to the desired position for the new point on the line. Click the left mouse button to set a point. A point is added to the line.



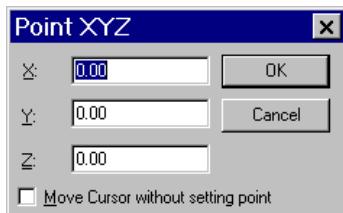
Point XYZ Command

Menu:	POINT
Menu Command:	POINT XYZ
Shortcut Key:	: (colon)

This command is used to set a point by specifying the X, Y, and Z coordinates of that point.

Using the Command

To set a point with this command, enter the coordinates in the X, Y, and Z boxes and click on OK. You can also move the cursor to the new location without setting a point there by selecting that option.



Polygon (Center-Vertex) Command

Menu: DRAW
 Submenu: PLANES
 Menu Command: POLYGON (CENTER-VERTEX)



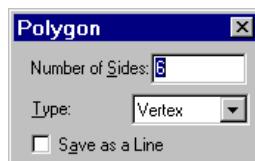
Toolbox Icon:

Point 1: Center of the surface
 Point 2: A corner of the surface
 Point 3: An orientation point to align the surface in space

The Polygon (Center-Vertex) command draws a regular polygon—that is, a convex shape in which each side is the same length.

Using the Command

Set two points to designate one side of the polygon. A third point can be used to determine the plane on which the polygon lies. The number of sides can be specified in the dialog box.



If you select **SAVE AS LINE**, the polygon will be saved as a line entity instead of a plane.

You can choose to draw the polygon either from center to a vertex or to the midpoint on one of the sides.

Example: Draw a polygon with a specific center point.

Select the POLYGON (CENTER-VERTEX) command and set the first point where you want the center of the object. Set the second point the distance that you want the vertex from the center. You can set a third point to determine the plane of the polygon.

Polygon (Edge) Command

Menu: DRAW
 Submenu: PLANES
 Menu Command: POLYGON (EDGE)



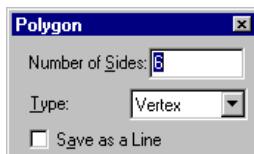
Toolbox Icon:

Point 1: Start of one side of the polygon
 Point 2: End of one side of the polygon
 Point 3: Orientation of the polygon

The Polygon (Edge) command draws a regular polygon—that is, a convex shape in which each side is the same length. The number of sides can be entered in the dialog box.

Using the Command

Set two points to designate one side of the polygon. A third point can be used to determine the plane on which the polygon lies.



If you select the **SAVE AS A LINE** option, the polygon will be saved as a line instead of a plane entity.

This command is similar to the Polygon (Center-Vertex) command. With the Polygon (Edge) command, you set two points for one edge of the regular polygon. With the Polygon (Center-Vertex) command, you set a point at the center and a point at a vertex of the polygon.

Example: Draw a polygon in your drawing.

Select the **POLYGON (EDGE)** command. Set a point for the beginning of one of the sides. Then move the cursor away from Point 1. A rubber-band polygon will be drawn using the cursor position as Point 2. When the polygon is the desired size, set Point 2. Hold down **Ctrl** while moving the mouse up or down and the polygon will swing on the axis of the first two points. When the polygon lies in the desired plane, set the last point.

Polygon Selection Command

Menu:	EDIT
Menu Command:	POLYGON SELECTION

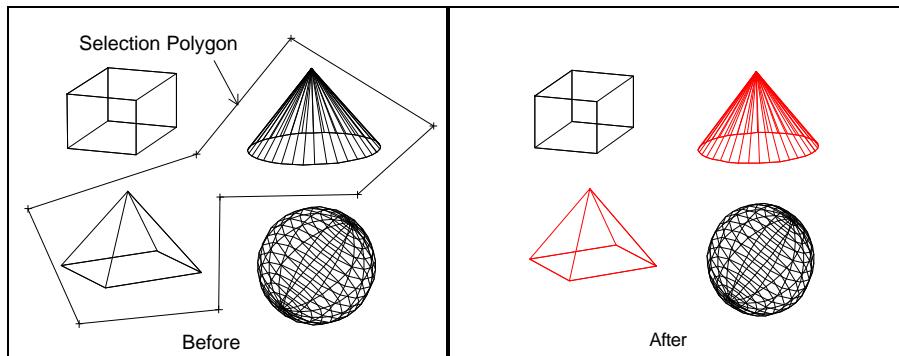
In 2-D Selection Mode and 3-D Selection Mode, the Polygon Selection command works like a 2-D selection rectangle. The selection polygon can consist of as many points and sides as you need. This allows you to define a more specific area for selection.

Using the Command

Select the **POLYGON SELECTION** command from the **EDIT** menu. Set points in your drawing forming a polygon around the entities you wish to select. At least three points must be set to form a valid polygon. If boundaries of the polygon overlap one another, entities encompassed by the resulting area will not be selected. Press **Enter**. The entities which were inside the polygon are selected.

Example: Select a pyramid and a cone.

Draw a box toward the left side of the screen. Draw a pyramid right below the box. Draw a cone on the right side of the box. Draw a sphere below the cone. Choose **POLYGON SELECTION**. Set points defining a polygon around the pyramid and the cone. Be sure that only the cone and the pyramid are completely inside the polygon. Press **Enter**. The pyramid and the cone are selected. The box and the sphere are not.



Preset Point Mode Command

Menu: **OPTIONS**

Menu Command: **PRESET POINT MODE**

Toolbox Icon:

When Preset Point Mode is active, the points needed for a drawing can be set before the commands are chosen. This allows you to set points in precise locations without seeing a rubber-band preview of the entity you are drawing.

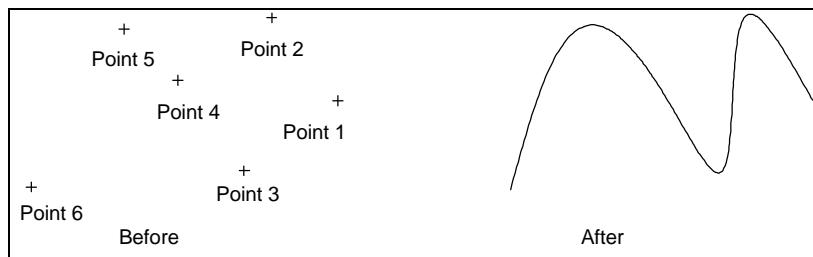
Using the Command

Choose PRESET POINT MODE from the OPTIONS menu. A check will appear in the menu beside Preset Point Mode when the command is enabled. Set the points in your drawing. Select a drawing command. The entity you selected will be drawn using the points you set previously.

If more points have been set than are necessary for the chosen entity, DesignCAD will use only as many points as needed, in the order in which they were set. The remaining points will be removed. For instance, if you set four points and choose the box command, the first two points set will be used to draw the box. The third and fourth points will disappear.

Example: Draw a curve.

Choose PRESET POINT MODE from the OPTIONS menu. Set six points in the drawing. Select the CURVE command. Press **Enter**. A curve will be drawn through the six points you set.



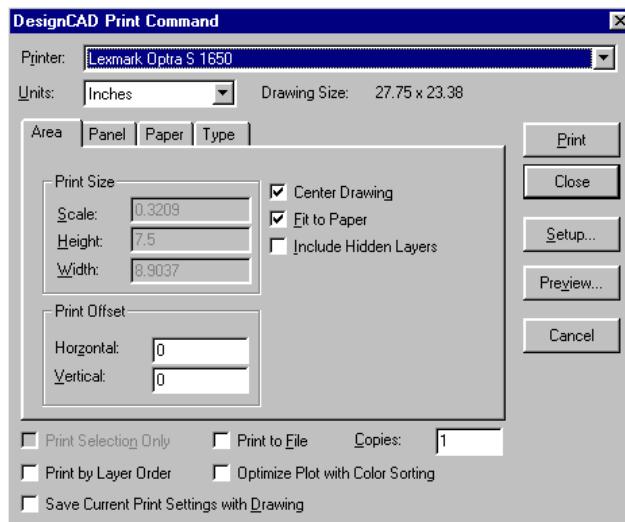
Print Command

Menu:	FILE
Menu Command:	PRINT
Shortcut Key:	Ctrl+P
Toolbox Icon:	

The Print command outputs your drawing to a printer or plotter. The drawing is printed at the view and perspective of the current view.

Using the Command

By default, clicking the Print icon automatically sends the current view to the currently selected printer and prints it at the largest size possible on a single sheet of paper. The Enable Direct Print from Toolbar option in the General Options folder can be disabled to make clicking on the Print icon bring up the Print command dialog box instead. To specify different printer settings, select the PRINT command from the FILE menu or press **Ctrl+P**, the shortcut key. The Print dialog appears.



Print Options

DesignCAD offers many options, but you don't have to use them all. If you want to print your drawing on a single sheet of paper and the maximum size, just select FIT TO PAPER and click on PRINT. The other options can be used to print to scale, rotate the drawing 90 degrees, set the margins, and so forth.

Selecting the Printer

Select the printer for the pending print job from the list of available printers by clicking on the DOWN ARROW to the right of the PRINTER: label and clicking on the printer name. If you would like to see the properties for the selected printer, click on the Setup button.

Note: If you do not see the printer you would like to use for the print job in the list of available printers, make sure the printer is installed correctly in Windows. If you have problems, consult your Windows documentation and your printer manual. Make sure you have not only the correct driver installed, but also that it is the most recent driver from the manufacturer.

Units

This is the unit of measurement for the paper on the printer. Don't confuse this item with Drawing Units. Drawing Units are the units of measurement of the drawing itself, *not* the printer.

Drawing Size

This represents the size of the box that would just fit around the current view if the drawing were projected onto a flat surface. The size is given in DesignCAD Drawing Units.

Area

This group of options let you control the size of your printed drawing.

Scale

This number is a ratio. It represents the number of paper units that will be used to print one Drawing Unit. Suppose, for example, that you are printing the front view of a 2x2 box, and you have chosen "inches" for the paper units. If you set Scale to 1.0, then one inch of paper will be used for each Drawing Unit and the box will be printed out at two inches by two inches on the paper. If you set Scale to 2.0, then two inches of paper will be used to print each Drawing Unit, and the box will be printed at four inches by four inches. With a Scale of 0.25, the box will be printed at 0.5 inches by 0.5 inches.

$$\text{Scale} = \text{Paper Units} / \text{Drawing Units}$$

Height

This is the height of the printed drawing on the paper if printed at the current scale. If you are printing multiple panels, this number is the total height of the drawing across all panels.

Width

This option reports the width of the printed drawing at the current scale. If you are printing multiple panels, this number is the overall width of the drawing once the panels are assembled.

Center Drawing

This centers the drawing on the paper.

Fit to Paper

This outputs the drawing on a single page in the largest possible size. If this option is selected, the Scale, Height, and Width cannot be entered.

Include Hidden Layers

This option takes the entire drawing (including hidden layers and unselected objects) into consideration when determining the scale and center position of the printed drawing. This is an easy way to print transparencies.

Horizontal

This option specifies how far the drawing will be printed from the left edge of the leftmost piece of paper for the print job. Positive values move the position of the printed image to the right.

Vertical

This option specifies how far the drawing will be printed from the top edge of the topmost piece of paper for the print job. Positive values move the position of the printed image down.

Panel

This group of options gives you information about how multiple panels will be handled. If a drawing is scaled too large to fit on a single page, DesignCAD will print the portions of the drawing on separate sheets of paper which can then be assembled. For example if a drawing is printed in four panels, then each panel would contain one-quarter of the drawing. This makes it possible to print drawings at a large scale with standard paper sizes.

Panels: 1 (1x1)

This line displays the total number of panels to be printed, and how many across by how many down.

Mark

You can choose whether to mark the boundaries of the individual panels. Your choices are NONE, CORNER MARK, and OUTLINE BOX.

NONE: No panel marking is done.

CORNER MARK: A registration mark is printed at each of the corners of the panels. This makes it easy to align the panels after printing.

OUTLINE BOX: Boxes are drawn around each panel to indicate margin locations. Each panel can be trimmed to the edge of these boxes so that when the panels are assembled edge-to-edge, the alignment of the panels will be correct. This way no paper overlap will interfere with the image.

Mark Panel Number

If this box is checked, each panel will be numbered. This is useful in arranging the panels correctly when assembling the finished printout of the drawing.

Enable Panel Overlap

Use this option to ease the assembly of paneled drawings. This option creates a specified image overlap between the panels. This eliminates the need for precise panel trimming before assembly.

Overlap Size

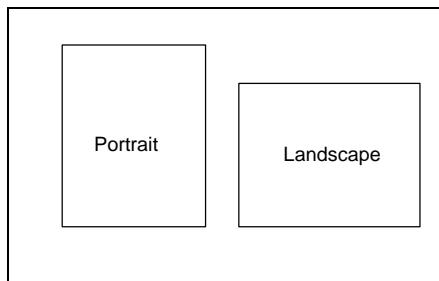
This box is used to specify the amount of overlap DesignCAD prints on a panel when the Panel Overlap option is used. The Overlap Size is applied to each side of every panel that overlaps another panel.

Paper

This group of options lets you choose your paper orientation and margins.

Orientation

Displays the currently selected orientation. A drawing can be printed using Portrait or Landscape orientation. It can be changed by clicking the SETUP button in the Print Command dialog box.

**Paper Size**

This displays the currently selected paper size. It can be changed by clicking the SETUP button in the Print Command dialog box.

Margin

This option determines the size of the page margins.

- TOP: The size of the top margin.
- BOTTOM: The size of the bottom margin.
- LEFT: The size of the left margin.
- RIGHT: The size of the right margin.

Type

The TYPE options let you print your drawing in several forms:

- WIRE FRAME
- QUICK SHADE
- SMOOTH SHADE
- HIDE (prints the drawing using hidden line removal)

Dimension

This option prints dimensions in the drawing.

Text

This option prints text regardless of whether it is on the near side, inside, or far side of a shaded solid. This option also shows text in views in which the hidden lines have been removed.

Line

This option prints lines regardless of whether they are on the near side, inside, or far side of a shaded solid. This option also shows all lines in views that have the hidden lines removed.

Arrow

This option prints arrows regardless of whether they are on the near side, inside, or far side of a shaded solid. This option also shows all arrows in views that have the hidden lines removed.

Pen Width

This option can be used to increase the print width of the lines in a drawing. With some high resolution printers, the zero width lines are difficult to see or do not print as complete lines. This option alleviates some problems with printing zero width lines.

Print as Black and White

Some colors print very lightly on black and white printers. When you are not using a color printer to print your color drawings, check this option to print all drawing entities as if they were black.

Render Shaded Image with OpenGL

This option tells DesignCAD to use OpenGL shading to print the drawing.

Pen Plot Hidden Line

This option tells DesignCAD to send the print/plot information necessary for a hidden line removal to a pen plotter.

Print Display Grid

If you have used a display grid in your drawing and the drawing is in 2-D Mode, this option tells DesignCAD whether or not to print the Display Grid with the drawing.

Print Selection Only

If this box is checked, only the currently selected object or objects will be printed. If no objects are selected, this box is not available.

Print by Layer Order

The Print by Layer Order option tells DesignCAD to send the drawing to a plotter one layer at a time.

Note: The status of the Print by Layer Order option is not saved in the drawing file, thus you will need to reselect it when you get ready to plot the drawing in subsequent drawing sessions.

Save Current Print Settings with Drawing

Saves the options set in the Print dialog so that the drawing can be printed again without resetting the options. This option also makes it possible to print the drawing with the same options using direct print from the Toolbar.

Print to File

This option can be used to send the print information to a file rather than to the actual printer. This is convenient if you need to print or plot your drawing at another location. It can also be used to transfer the drawing image to other applications.

Optimize Plot with Color Sorting

The Optimize Plot with Color Sorting option tells DesignCAD to send the drawing to a plotter one color at a time.

Note: The status of the Optimize Plot with Color Sorting option is not saved in the drawing file, thus you will need to reselect it when you get ready to plot the drawing in subsequent drawing sessions.

Copies

Enter in this box the number of copies to be printed.

Print

This command starts the printing process.

Close

Clicking this button saves the changes you have made to the options in the Print dialog box, closes the Print dialog box, and returns you to your drawing.

Setup

Displays the options for the currently selected printer.

Preview

Clicking this button brings up the Print Preview screen, which shows exactly how your drawing will look on paper.

For more details on the Print Preview Options, see the Print Preview entry in the Command Reference section of this Reference Manual.

Cancel

Clicking this button disregards any changes you have made to the options in the Print dialog box, closes the Print dialog box, and returns you to your drawing.

Example: Create or load a DesignCAD drawing and print it.

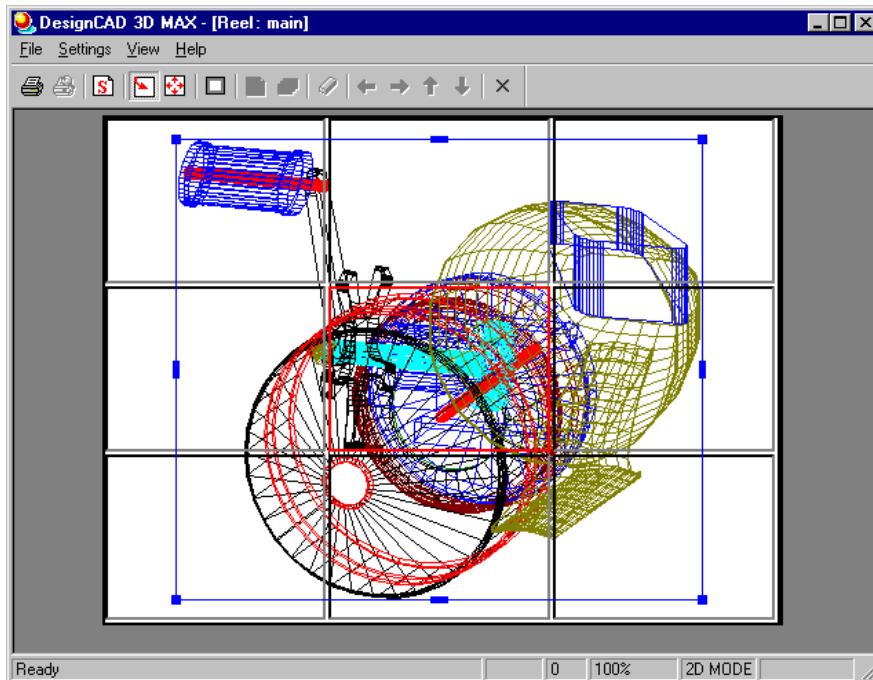
Create or load a drawing. Select the PRINT command from the FILE menu. In the Print Command dialog box, make sure the CENTER DRAWING and FIT TO PAPER checkboxes are checked. Click on the TYPE tab and then select WIRE FRAME in the top list box. For this example leave the rest of the

options at their defaults, but be aware that you can adjust them to your liking. Click the PRINT button. Your drawing will be printed on the printer currently selected in the PRINTER list box.

Print Preview Command

Menu: FILE
 Menu Command: PRINT PREVIEW
 Toolbox Icon: 

The Print Preview command shows exactly how your drawing is going to look on paper. There are also several options that can be adjusted from the Print Preview screen.



Print

Prints the drawing.

Change Printer Setting

Brings up the Print Command dialog box, making it possible to change the settings for the print job. From this dialog box, pressing **Enter** will print the drawing. To get back to the Print Preview screen, click the CLOSE button.

Center Drawing

If the drawing fits on a single piece of paper, this command will center the drawing on the paper. If it is a multi-paneled drawing, this command centers the drawing so that there is the

same amount of white space above and below the drawing and the same amount of white space on each side of the drawing.

Fit to Paper

Outputs the drawing on a single piece of paper in the largest size possible.

Print Region

Lets you define a region of the drawing to be printed. Click the PRINT REGION icon in the Print Preview toolbox. Move the mouse to one corner of the region. Hold down the left button and drag a rectangle around the region. Release the mouse button at the opposite corner.

Show Single Panel

Shows only the currently selected panel, enlarged to fit the screen.

Show All Panels

Shows all panels of the drawing on the screen.

Mark/Unmark Skipped Panel

Works as a toggle to select or deselect individual panels and determine whether or not they will be printed. Panels that will not be printed are grayed out.

Left Page

In a multi-paneled view, selects the panel to the left of the currently selected panel. In a single-paneled view, changes the view so that the panel being viewed is the one to the left of the panel currently being viewed.

Right Page

In a multi-paneled view, selects the panel to the right of the currently selected panel. In a single-paneled view, changes the view so that the panel being viewed is the one to the right of the panel currently being viewed.

Up Page

In a multi-paneled view, selects the panel directly above the currently selected panel. In a single-paneled view, changes the view so that the panel being viewed is the one directly above the panel currently being viewed.

Down Page

In a multi-paneled view, selects the panel directly below the currently selected panel. In a single-paneled view, changes the view so that the panel being viewed is the one directly below the panel currently being viewed.

Close Print Preview

Closes the Print Preview dialog box and returns to the Print Command dialog box.

When the entire drawing is shown, either on a single sheet or in panels, there are blue scaling and moving nodes around the drawing.

The small blue box at each corner is a scaling node and can be used to scale the drawing. To scale the drawing, click on one of the scaling nodes and hold down the mouse button. Move the mouse until the drawing is scaled to the desired size, then release the mouse button.

The small blue box on the top, bottom and each side of the drawing is a moving node. To move the drawing, simply click and hold on one of the moving nodes or one of the blue lines around the drawing. Use the mouse to move the drawing to the desired location, then release the mouse button. If you move the drawing past the edge of the pages shown, DesignCAD automatically adds pages as needed for the print job.

Pullout Command

Menu: DIMENSION

Submenu: INFO

Menu Command: PULLOUT



Toolbox Icon:

Point 1: Arrowhead

Point 2-n: Path of arrow. The last point marks the text location.

The Pullout command inserts arrows and descriptive text into a drawing.

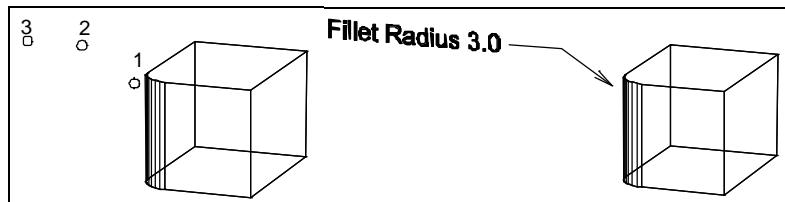
Using the Command

Choose the PULLOUT command from the INFO submenu of the DIMENSION menu. Enter the text information in the TEXT: box, and click on the ARROWHEAD button for a choice of arrowhead styles or the TEXT button for a choice of text options. Set a point for the point of the arrow, and one or more other points for the body of the arrow. The last point is the location of the text.



Example: Adding a pullout to describe a fillet radius

Run the Pullout command from the INFO submenu of the Dimension menu. Enter the text "Fillet Radius 3.0" in the Text box. Set point 1 for the arrowhead, point 2 for a bend in the arrow, and point 3 for the text location. Press **Enter** to end the command.



See Also: *Arrow Command, Balloon Command*

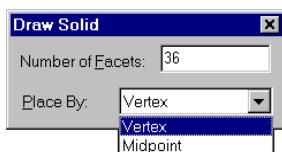
Pyramid Command

Menu: SOLIDS
 Menu Command: PYRAMID
 Toolbox Icon: 
 Point 1: Center of the pyramid's base
 Point 2: Radius of the pyramid
 Point 3: Height of the pyramid

The Pyramid command draws a solid pyramid.

Using the Command

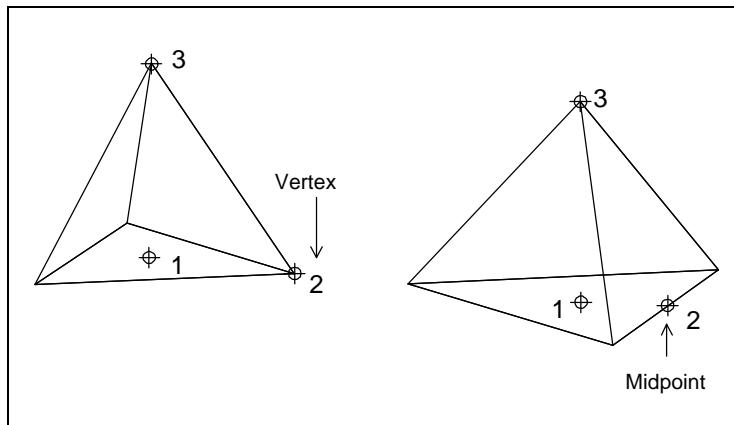
You can specify the number of sides or facets along the longitude and the latitude of the pyramid. Enter the number of sides for the pyramid in the NUMBER OF FACETS: box in the dialog box.



You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Point 2. If you choose VERTEX, the base of the pyramid is inscribed by a circle of that radius. If you choose MIDPOINT, the base of the pyramid circumscribes a circle of that radius. This is normally not significant, but it can be important for some precision drawings.

Example: Draw a pyramid in your drawing.

Select the PYRAMID command. Set a point for the center of the pyramid's base. Move the cursor out along the Y axis and set the second point for the radius of the base. Next, set a point for the height of the pyramid.



Quarter Circle Command

Menu: DRAW
 Submenu: ARCS
 Menu Command: QUARTER CIRCLE
 Shortcut Key: (
 Toolbox Icon:
 Point 1: Beginning of the quarter circle.
 Point 2: Endpoint for the quarter circle.

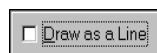
The Quarter Circle command draws a quarter circle, or 90 degree arc, between two points.

Using the Command

Choose the QUARTER CIRCLE command in the Toolbox. Set a point for the beginning of the quarter circle. After the first point is set, a rubber-band circle shows how the quarter circle will be drawn. Set a point for the end of the quarter circle. A quarter circle, or 90 degree arc, will be drawn counterclockwise from the first point to the second point.

There are two choices for the format of the quarter circle, which you can set in the dialog box:

- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Draws line segments that follow the shape of an quarter circle.



The Arc format saves the quarter circle as an Arc entity. The Line format saves the quarter circle as a series of short line segments. You should normally use the Arc format because it is more efficient and precise. You can use the Line format if you need to distort the quarter circle by scaling or stretching.

Record Options Command

Menu: TOOLS
 Menu Command: RECORD OPTIONS

The Record Options command can be activated while the macro is being recorded. This command allows you to save and change options within the macro.



The Record Options command only allows you to change "Save" macro options:

Record Starting Point

Choose this box to record a starting point for the macro. If this option is used when the macro is recorded, a starting point can be set when the macro is executed.

Record Layer

Choose this box to record layer information such as layer color, layer name, and current layer with the macro.

Record Command Parameters

Choose this box to record all drawing options including color, line type, dimension options, and layer options. However, some information entered in the dialog box or in the Text Block dialog box is not recorded with this command.

Record Points

Choose this box to record any points set within the macro.

Record Color

Choose this box to record the drawing's current color with the macro.

Record Line Style

Choose this box to record the drawing's current line style with the macro.

Note: You can also assign Toolbox buttons to the macros you create. See the Toolbox Options entry in the "Command Reference" section of this manual for more information.

Redo Command

Menu:	EDIT
Menu Command:	REDO
Shortcut Key:	Ctrl+Y
Toolbar Icon:	

The Redo command reverses any action performed by the Undo command. It is only available after you have used the Undo command in the current drawing session.

Using the Command

Press the shortcut key, click the Toolbar icon or choose the command in the EDIT menu. The preceding action is immediately reversed.

Example: Replace a line accidentally erased with the Undo command.

Select the REDO command. The line is redrawn in the same location.

See Also: Undo Command

Refresh Command

Toolbox Icon:



The Refresh command refreshes the active drawing window. It cleans up images, such as cursor cross hairs, that have been left on the screen. Choose this icon when you do not want to redraw the screen but only refresh it.

Using the Command

Make sure that the window to be refreshed is active. Click on the REFRESH icon. Any remnants of the cursor used in previous drawing commands are erased and blank spots caused by selection handles are restored to their former appearance.

Note: When working with large, detailed drawings, regenerating the entire screen takes more time than refreshing it. If you just want to clean up the effects of previous cursor locations or selection handles, use the Refresh command.

See Also: *Regenerate Command*

Regenerate Command

Menu:

VIEW

Menu Command:

REGENERATE

Shortcut Key:

Ctrl+R

Toolbox Icon:



The Regenerate command redraws the entire drawing in wireframe form. This erases any shaded or hidden-line areas of the screen. This command affects only the active view window.

Using the Command

Choose the Regenerate command. The program immediately begins to redraw the entire drawing in wireframe form.

Example: Return a shaded image to wireframe format.

Select the REGENERATE command. The object is redrawn in wireframe.

See Also: *Regenerate All Command, Regenerate Double Line Entities Command*

Regenerate All Command

Menu:

VIEW

Menu Command:

REGENERATE ALL

Shortcut Key:

Ctrl+Shift+R

This command redraws the entire drawing in wireframe form in every open view window. It erases any shaded or hidden line areas of the screen.

Using the Command

Choose the Regenerate All command. All objects on the drawing screen, in all views, are redrawn in wireframe form.

Example: Redraw an object in wireframe in all views.

Select the REGENERATE ALL command. The object is redrawn in wireframe format in all views.

See Also: *Regenerate Command, Regenerate Double Line Entities Command*

Regenerate Double Line Entities Command

Menu:	VIEW
Menu Command:	REGENERATE DOUBLE LINE ENTITIES
Toolbox Icon:	

The Regenerate Double Line Entities command redraws all of the double line entities in the drawing. Sometimes deleting a line or entity that intersected a double entity will give the appearance that part of the double entity was erased. The Regenerate Double Line Entities command refreshes the double entities, removing any apparent gaps.

Using the Command

Choose the Regenerate Double Line Entities command. The program immediately begins to redraw all of the double line entities in the drawing.

See Also: *Regenerate Command*

Remove Menu Item Command

Menu:	TOOLS
Submenu:	DIGITIZER
Menu Command:	REMOVE MENU ITEM
Point 1:	Point inside the menu area

The Remove Menu Item command removes a command from an existing digitizer menu.

Using the Command

Open the digitizer menu to be changed. Choose the REMOVE MENU ITEM command from the DIGITIZER submenu of the TOOLS menu. Set a point in the area occupied by the command you want removed. The command is removed from the digitizer menu.

See Also: *Add Menu Item Command, Close Digitizer Menu Command, Create Digitizer Menu Command, Load Digitizer Menu Command, Remove Menu Item Command*

Reset Working Plane Command

Menu:	VIEW
Submenu:	WORKING PLANE
Menu Command:	RESET WORKING PLANE

The Reset Working Plane command reverses a Set Working Plane command. It restores the original coordinate system to the drawing, deleting any reference to the temporary coordinate axes set up by the Set Working Plane command.

Using the Command

Choose the **RESET WORKING PLANE** command from the **WORKING PLANE** submenu of the **VIEW** menu. Like the Undo command, this one reverses a previous action, in this case restoring the original coordinate system to the drawing.

See Also: ***Set Working Plane Command***

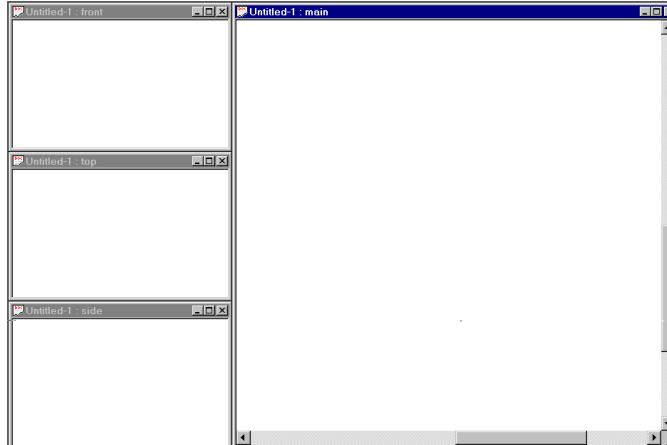
Restore DesignCAD Tile Command

Menu:	WINDOW
Submenu:	DESIGNCAD TILE SETTINGS
Menu Command:	RESTORE DESIGNCAD TILE

The Restore DesignCAD Tile command changes the DesignCAD Tile setting back to its default after it has been changed with the Set As DesignCAD Tile command. The Restore DesignCAD Tile command resets the DesignCAD Tile setting to its default view configuration: the Perspective view in the large window on the right side of the screen, and the Front, Top, and Side views stacked vertically on the left.

Using the Command

After changing the DesignCAD Tile setting with the **SET AS DESIGNCAD TILE** command, choose the **RESTORE DESIGNCAD TILE** command from the **DESIGNCAD TILE SETTINGS** submenu in the **WINDOW** menu. The program arranges the windows according to the default DesignCAD Tile setting with Perspective, Front, Top, and Side views.



See Also: ***DesignCAD Tile Command***, ***Set As DesignCAD Tile Command***

Rotate Command

Menu: EDIT
 Submenu: SELECTION EDIT
 Menu Command: ROTATE
 Shortcut Key: R

Toolbox Icon: 

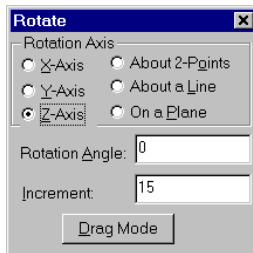
Point 1: Location of rotation axis

Point 2: Second point on rotation axis (2-Point rotation only)

This command can be used to rotate a selected object or group of objects.

Using the Command

Select the object or objects you want to rotate, and then choose the ROTATE command. The selection can be rotated on any axis at any angle. The dialog box displays several options for the axis of rotation:



X: Rotate about the selection handle on the X axis.

Y: Rotate about the selection handle on the Y axis.

Z: Rotate about the selection handle on the Z axis.

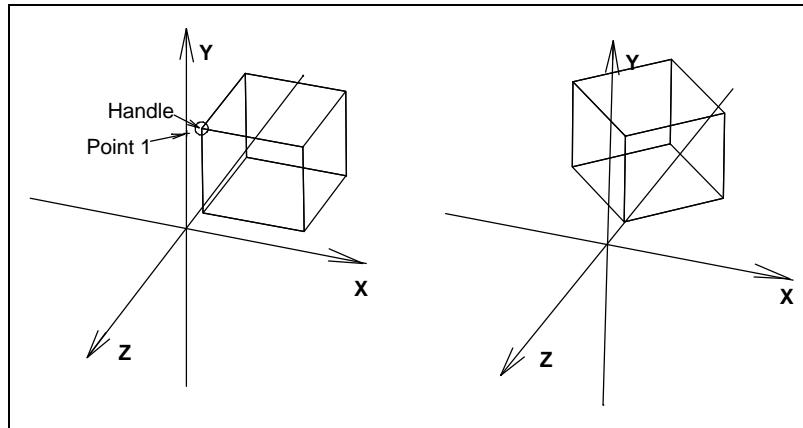
ABOUT 2-POINTS: Set two points for the axis of rotation.

ABOUT A LINE: Set a point on the line that is to be the axis of rotation.

ON A PLANE: Rotate on a plane (about the axis of plane's normal). Set a point on the plane and one for the rotation center.

Example: Rotate a box 45 degrees on the Y axis.

Select a box. Choose the ROTATE command. Enter 45 in the ROTATE ANGLE field in the dialog box. Select the Y option for ROTATE AXIS. Set a point on the box's handle. The box rotates 45 degrees on the Y axis.



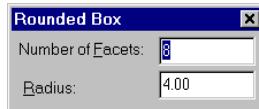
Rounded Box Command

Menu: SOLIDS
 Menu Command: ROUNDED BOX
 Toolbox Icon: 
 Point 1: First corner of the box
 Point 2: Opposite corner of the box

The Rounded Box command draws a solid box with filleted (rounded) corners and edges.

Using the Command

Select the ROUNDED BOX command. You can specify the number of facets (or "faces") for the corners and edges and the radius of these facets. Click in the ROUND BOX FACETS: box in the dialog box and enter number of facets to be drawn for each corner and edge. In the ROUND BOX RADIUS: box and enter the radius for the rounded corners and edges.



When you have entered the values for the Round Box Facets: and Round Box Radius: options press the **Enter** key. Set a point for the first corner of the rounded box. Move the cursor along the X, Y, and Z axes. A rubber-band box appears to help you determine the size of your box. When the rubber-band box is the desired size, set a second point for the opposite side of the rounded box. The rounded box is drawn at the size specified by the two points set and by the values entered for the options in the dialog box.

Example: Draw a rounded box in your drawing.

Select the ROUNDED BOX command. Enter 8 in the ROUND BOX FACETS: box. Enter 4 in the ROUND BOX RADIUS: box. Set a point for the first corner of the rounded box. Move the cursor up and to the right of the first one, noticing the rubber-band square being drawn. When the square is the

size you want, move the cursor out along the Z-axis by holding down **Ctrl+Shift** and moving the mouse up. The square turns into a 3-D box. When the rubber-band box is the desired size, set a point. A box is drawn with filleted corners and edges. Each fillet has a radius of 4 and is drawn using 8 facets.

Ruler Command

Menu:	VIEW
Menu Command:	RULER

The Ruler command displays or hides vertical and horizontal rulers beside the drawing window when in 2-D Mode.

Using the Command

While in 2-D Mode, choose the RULER command from the VIEW menu. This is a toggle command: select to place a check mark beside the command to turn it on; select again to remove the check mark and turn the command off.

Hint: You can set ruler divisions with the View Options command. To change ruler settings, choose the OPTIONS command from the OPTIONS menu and click the VIEW OPTIONS tab. Click the RULER SETTINGS button.

Run Executable Command

Menu:	TOOLS
Menu Command:	RUN EXECUTABLE

The Run Executable command lets you run an executable from within DesignCAD.

Using the Command

Choose the Run Executable command. The Run Add-On box appears. Select or enter the name of the file and its location in the FILE NAME and LOOK IN boxes respectively. When you have supplied the information, click the OPEN button. The executable initializes.

To stop an executable, select the program's Exit command.

Example: Run an executable from within DesignCAD.

Select the RUN EXECUTABLE command. Select the appropriate directory and file name in the dialog box. Click OPEN.

Run Walk Command

Menu:	WALK
Menu Command:	RUN WALK

The Run Walk command begins the playback of a walk template created with Walk Through Mode.

Using the Command

After creating a walk through template in Walk Through Mode, select the RUN WALK command. The walk through begins to play.

Save Command

Menu:	FILE
Menu Command:	SAVE
Shortcut Key:	Ctrl+S
Toolbox Icon:	

The Save command saves the active drawing to disk. If the drawing is new and has not yet been saved, you will be asked to name the file you want to save.

Using the Command

Choose the SAVE command. If this is your first time to save the drawing, the Save As box appears. Enter the name of the file you want to save and where you want to save it. Then click OK. If you have saved the drawing before, the Save command saves all changes to the drawing since the last time you chose the Save command.

The drawing handles are automatically placed at the lower-left, lower-right, and upper-right-front of the drawing. The Set Drawing Handles command can be used to specify the specific handle locations for the drawing.

See Also: *Save As Command*

Save Animation Template Command

Menu:	ANIMATION
Menu Command:	SAVE ANIMATION TEMPLATE

The Save Animation Template command saves an animation that has been produced in Animation Mode.

Using the Command

After producing an animation in Animation Mode, save your animation so that it can be opened and viewed later. Select the SAVE ANIMATION TEMPLATE command from the ANIMATION menu. Enter a name for the animation in the dialog box and then click OK.

Note: The animation template is saved as a part of the drawing file it was created with and can only be opened with the Load Animation Template command if the drawing file is open in DesignCAD.

See Also: *Animation Mode Command, Load Animation Template Command*

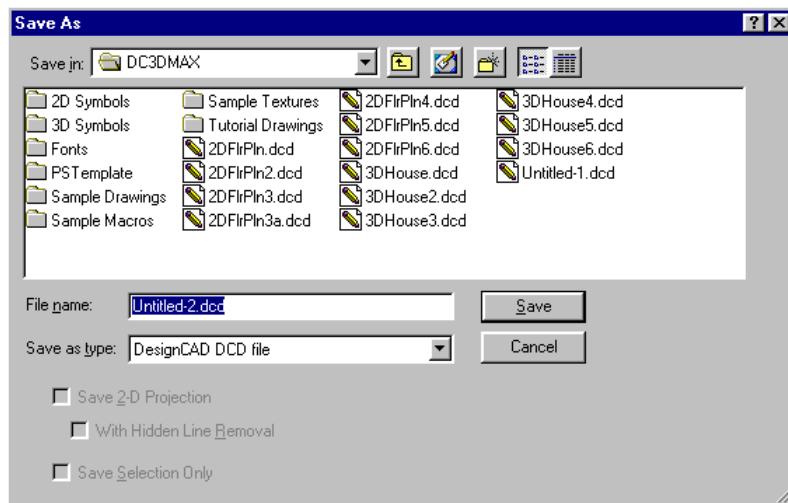
Save As Command

Menu: FILE
 Menu Command: SAVE AS

The Save As command saves the active drawing to disk.

Using the Command

Choose the SAVE AS command. The Save As box appears. In the File Name box enter or select the name of the file to save. In the Save In box enter the location where you want to store the drawing. If you want to save the drawing as a file type different than the one you're working in, select the type in the FILES OF TYPE box. Then click OK.



Save 2-D Projection

Choose this option to save your 3-D drawing as a 2-D projection, so it can be opened as a 2-D drawing.

With Hidden Line Removal

This option is only available if you have checked the Save 2-D Projection option. This option will remove all hidden lines from the resulting 2-D file.

Save Selection Only

After selecting the portion of the drawing to be saved as a separate file, select the SAVE AS command and check the SAVE SELECTION ONLY option to save the selected objects as a separate file. Be sure to change the filename for the resulting file, so you don't overwrite the original.

Save Current Layout as Template Command

Menu: LAYOUT
 Menu Command: SAVE CURRENT LAYOUT AS TEMPLATE
 Toolbox Icon:

The Save Current Layout as Template command saves the current Paper Space Template, including all the View Frames and the options set for each.

Using the Command

While in Paper Space Mode, choose the **SAVE CURRENT LAYOUT AS TEMPLATE** command from the **LAYOUT** menu. The command can also be selected by clicking on the Save Current Layout as Template icon in the Paper Space Mode dialog box. Specify the name of the template and the location where you want to save it in the Paper Space Template dialog box. Click **OK**. The Paper Space Template is saved. Saving the template saves every View Frame's location and size, along with the specific options set for each.

Note: The Save Current Layout as Template command is only available when DesignCAD is in Paper Space Mode. The Layout menu appears in the Command Menu when Paper Space Mode is enabled.

Templates you create and save can be loaded using the Load Paper Space Template command.

Save Current View Command

Menu:	VIEW
Menu Command:	SAVE CURRENT VIEW

The Save Current View command saves the current view settings as a custom view selection in the Viewing Toolbox.

Using the Command

Choose the **SAVE CURRENT VIEW** command. When the Save Current View box appears, enter the name of the view. The name stays in the view list until it is removed. (To remove it, click on the view name in the Viewing Toolbox and then press the **Del** key.)



Example: Create a drawing containing several objects.

Change the viewer position by clicking the viewer position buttons, or entering new angles in the view angle fields in the Viewing Toolbox.



When you have adjusted the view to your liking, select the **SAVE CURRENT VIEW** command. Enter a name for your custom view in the **CURRENT VIEW NAME** field and click **OK**. The view is saved under the selected name in the view box in the Viewing Toolbox. Next, remove the view by selecting the view and pressing the **Del** key.

Save Digitizer Menu Command

Menu: TOOLS
 Submenu: DIGITIZER
 Menu Command: SAVE DIGITIZER MENU
 Point 1: Point inside the digitizer menu

The Save Digitizer Menu command saves the current digitizer menu. This command saves the menu in .DGM file format.

Using the Command

After creating the digitizer menu, choose the SAVE DIGITIZER MENU command from the DIGITIZER submenu of the TOOLS menu. Set a point inside the menu area. The Save Digitizer Menu dialog box appears. Enter the name of the new digitizer menu in the FILE NAME box and click SAVE.

See Also: *Add Menu Item Command, Close Digitizer Menu Command, Create Digitizer Menu Command, Load Digitizer Menu Command, Remove Menu Item Command*

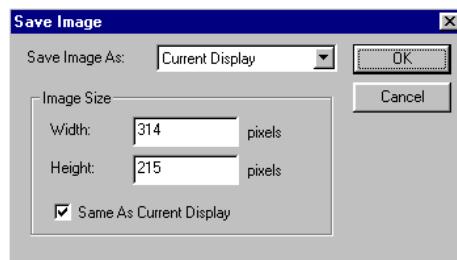
Save Image File Command

Menu: FILE
 Submenu: IMAGE
 Menu Command: SAVE IMAGE FILE
 Point 1: First corner of rectangular region to save
 Point 2: Opposite corner of rectangular region to save

The Save Image File command saves a screen image (or part of one) as a graphics file.

Using the Command

When you choose the SAVE IMAGE FILE command, an X-shaped cursor appears in the drawing window. To select a portion of the window to save in a common graphic format, set a point in two opposite corners of the region to be saved. To save the entire window as a graphics file, press **Enter** without setting any points. The Save Image dialog box appears



Save Image As

This option lets you save the image in the current display mode, wireframe, quick shading, smooth shading, or hidden line removal.

Image Size

Width

Sets the pixel width of the image to be saved.

Height

Sets the pixel height of the image to be saved.

Same as Current Display

This option resets the values for the width and height of the image to be saved.

OK

Clicking on this button saves the changes you have made and opens the Save Image File dialog box. Enter the name and location for the new graphic file in the Save Image File box and then click the OK button to save it.

The saved file can be used with other applications such as word processors and desktop publishing systems. This is a convenient way to transfer a shaded image to other applications.

Cancel

Clicking on this button disregards the changes you have made and returns you to your drawing.

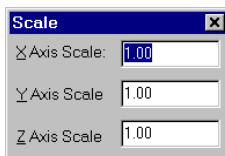
Scale Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	SCALE
Shortcut Key:	S
Toolbox Icon:	
Point 1:	Scale center

The Scale command is used to scale a selection along the X, Y, or Z axis. In other words, you can "stretch" the selected objects to make them taller, shorter, longer, wider, etc.

Using the Command

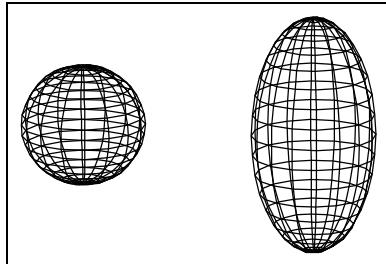
To use the command, select the object and choose the SCALE command. In the dialog box enter the scale factors for the x, y, and z directions.



For example, to make an object twice as tall with the same width and depth, you would enter **2** for the y scale factor and **1** for the x and z factors. When a selection is scaled, it is scaled about the selection handle.

Example: Rescale an object so that it is twice as large along the Y axis.

Select the object and choose the SCALE command. In the dialog box enter 1 for the x and z scales, and 2 for the y scale. Press **Enter**. The sphere is redrawn with the new scale factors.



Scan Image Command

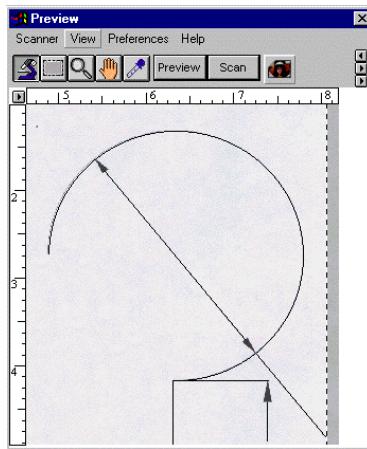
Menu:	FILE
Submenu:	IMAGE
Menu Command:	SCAN IMAGE

The Scan Image command gets an image from a TWAIN compatible scanner that has been specified with the Select Scanner Source command. A TWAIN scanner allows you to run the scanner directly from within DesignCAD. The image can later be converted into a vector image with the Auto Trace Bitmap command. This allows scanned artwork to be used in a DesignCAD drawing.

Using the Command

Place the image in the scanner. Make sure that the scanner has been specified with the Select Scanner Source command. Select the SCAN IMAGE command from the IMAGE submenu of the FILE menu.

Note: Scanners vary in their capabilities and features, and this is reflected in the dialog box that appears when the Scan Image command is used. Depending on the scanner's features, it may be possible to adjust the image's orientation, the direction of the scan, the resolution, etc. Refer to the scanner's software documentation or on-line Help for more information.



Use the scanner's own software to scan the image. The image is scanned and imported into DesignCAD. Use the Auto Trace Bitmap command in the TOOLS menu to trace the bitmap as a series of vectors.

If you are using a scanner that is not TWAIN compatible, you can still use scanned images in DesignCAD, but it takes more steps to process the image. Switch to the scanner software or driver to execute the scan, then return to DesignCAD. Once the image has been scanned as a bitmap, use the LOAD IMAGE FILE command in the IMAGE submenu of the FILE menu to bring the image into DesignCAD. After loading the image into DesignCAD, use the AUTO TRACE BITMAP command in the TOOLS menu to trace the bitmap as a series of vectors.

See Also: *Auto Trace Bitmap Command, Load Image File Command, Select Scanner Source Command*

Screen Configuration Load Command

Menu:	FILE
Submenu:	SCREEN CONFIGURATION
Menu Command:	LOAD

This command loads a previously saved screen configuration file and applies it to the current drawing.

Using the Command

Choose the SCREEN CONFIGURATION LOAD command. The OPEN VIEW FILE box appears. In the FILE NAME box enter the name of the view to load, and in the Look In box select the location of the view. Then click OPEN. The window configuration and view configuration for each window are loaded.

See Also: *Screen Configuration Save Command*

Screen Configuration Save Command

Menu:	FILE
Submenu:	SCREEN CONFIGURATION
Menu Command:	SAVE

This command saves the screen configuration of the current drawing. The window configuration and the view configuration for each window are saved with this command.

Using the Command

Choose the SCREEN CONFIGURATION SAVE command. The Save View File box appears. In the FILE NAME box enter the name of the view to save. In the STORE IN box select the location where you want to store the view. Then click SAVE.

This command is very useful if you have a screen configuration you use frequently.

See Also: *Screen Configuration Load Command*

Section Copy Command

Menu:	EDIT
Menu Command:	SECTION COPY
Point 1:	One corner of the cutoff box
Point 2:	Opposite corner of the cutoff box

The Section Copy command copies a section of the drawing that intersects a three-dimensional box to the Windows Clipboard, so it can be pasted into the same drawing, a different drawing, or a different application.

Using the Command

After choosing the SECTION COPY command, define the section by setting two points in opposite corners of a three-dimensional box. Any part of the drawing that is contained within the cutoff box boundary pasted to the Windows Clipboard.

Example: Copy part of a circle elsewhere in the drawing.

Select the SECTION COPY command. Set the points for the cutoff box so that a section of the circle is inside the box. After the cutoff box disappears, press **Ctrl+V** or select the PASTE command from the EDIT menu. A rubber-band box representing the scale of the object will follow the cursor as you move it around the screen. When you have selected a location for the copied section of the circle, set a point.

Section Cutoff Command

Menu:	EDIT
Submenu:	SECTION EDIT
Menu Command:	SECTION CUTOFF
Point 1:	One corner of the cutoff box
Point 2:	Opposite corner of the cutoff box

The Section Cutoff command cuts off a section of the drawing that intersects a three-dimensional box. It essentially separates everything inside the box from everything outside the box by cutting planes and breaking lines. Even though the drawing looks the same after the cutoff command, any entity crossing the cutoff box will be cut off.

Using the Command

After choosing the SECTION CUTOFF command, define the section by setting two points in opposite corners of a three-dimensional box. Any part of the drawing that crosses the cutoff box boundary will be cut off at the intersection.

It is convenient to use this command to cut out a section of an object so it can be moved or deleted.

Example: Convert a circle into two separate entities.

Select the SECTION CUTOFF command. Set the points for the cutoff box so that a section of the circle is inside the box. After the cutoff box disappears, set the cursor on the section of the circle that was inside the box and click the left mouse button to select it. Notice that only the section is highlighted. This is because it has been redefined as a separate entity. You can manipulate it using any command without affecting the rest of the circle.

Section Delete Command

Menu:	EDIT
Submenu:	SECTION EDIT
Menu Command:	SECTION DELETE
Shortcut Key:	D
Point 1:	First corner of the section to be deleted
Point 2:	Opposite corner of the section to be deleted

The Section Delete command is used to delete or erase a section of the drawing. A section is a three-dimensional box defined by two points in opposite corners.

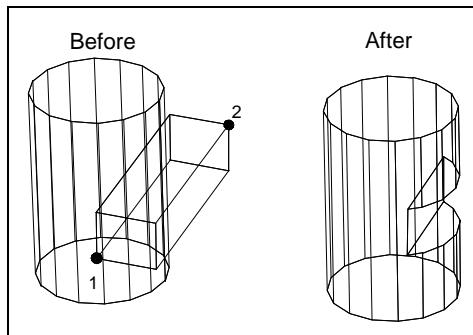
Using the Command

Choose the SECTION DELETE command. Define the section by setting two points in opposite corners of a three-dimensional box. Drawing entities within the section are deleted from the drawing. An entity partly inside the box is cut off, and only the part inside the box is erased.

Hint: This command is useful for cutting doorways and windows out of walls.

Example: Erase a rectangular section of a cylinder.

Select the SECTION DELETE command. Set the points of the bounding box so that part of the cylinder is enclosed in it. The cylinder will be redrawn without the section that was inside the bounding box.



See Also: *Section Cutoff Command*

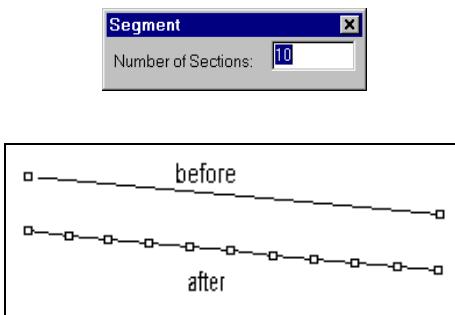
Segment Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: SEGMENT
 Point 1: Line or curve to be segmented

The Segment command allows you to divide a line or curve into a specified number of segments of equal length. Curves will automatically be subjected to Vector Convert before being segmented.

Using the Command

Choose the SEGMENT command from the EDIT menu. In the dialog box specify the NUMBER OF SECTIONS you want. Set a point on the line or curve to be segmented. The object is broken into the requested number of sections.



See Also: *Vector Convert Command*

Segregate Command

Menu: SOLIDS
 Menu Command: SEGREGATE
 Point 1: First Solid
 Point 2: Second Solid

The Segregate command is used to "cut out" the volume in common between two solids. This command is like the Exclusive Or command, but the common volume is cut out of the two solids and becomes a separate solid instead of just being subtracted from them.

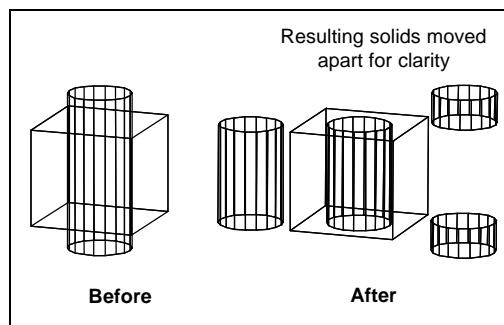
Using the Command

Choose the Segregate command. Set a point on each of the solids. The common volume between the two solids is cut out and becomes a third solid. The only portions that remain as the two original solids are portions that were exclusive to those solids before the Segregate command was executed.

Note: For best results, the solids should overlap rather than meet exactly at a face.

Example: Cut out the common volume from an intersecting box and cylinder.

Select the SEGREGATE command and set a point on each of the solids. DesignCAD cuts out the shared volume and redraws the remaining portions of the original solids.



Select All Command

Menu:	EDIT
Menu Command:	SELECT ALL
Shortcut Key:	Ctrl+A

The Select All command selects every editable object in the drawing. This makes it easy to manipulate all objects as a group rather than having to perform the same command on every object individually. Objects that reside in layers that have been defined as uneditable in the Layer Options folder, are not selected.

Using the Command

Choose the SELECT ALL command. Every entity that is in an editable layer in the drawing is automatically selected.

Hint: If you have a number of objects on the screen and need to select all but a few of them, choose the SELECT ALL command to select them all. Then depress the **Shift** key and click on the objects you don't want to be selected. This is much faster than selecting each object individually.

See Also: [2-D Selection Mode](#), [3-D Selection Mode](#)

Select Entity Command

Menu:	EDIT
Menu Command:	SELECT ENTITY

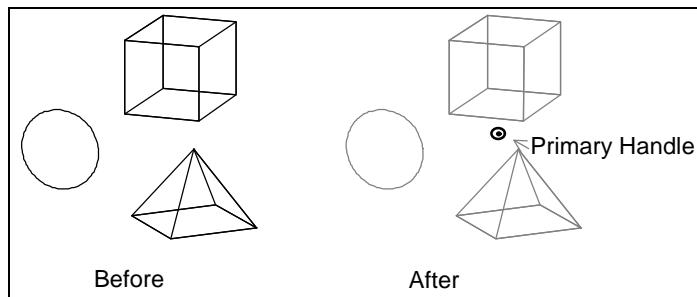
In Preset Point Mode, the points are set before the commands are chosen. Any time you click the mouse, a point is set. The Select Entity command tells DesignCAD that you want to select the entity on which you set a point.

Using the Command

In Preset Point Mode, set a point on each entity you wish to select. Choose the SELECT ENTITY command from the EDIT menu. Each entity on which a point was set will be selected.

Example: Select several entities.

Draw a box, a pyramid and a circle. Enable PRESET POINT MODE from the OPTIONS menu. Set a point on the box. Next set a point on the pyramid. Set another point on the circle. Choose SELECT ENTITY from the EDIT menu. The box, the pyramid and the circle will be selected.



Selection Filter Command

Menu:	EDIT
Menu Command:	SELECTION FILTER
Shortcut Key:	Shift+F

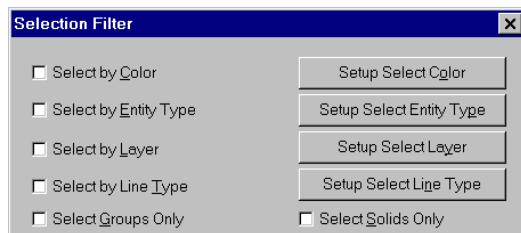
The Selection Filter command allows you to restrict the type of entities that are selected when a selection box is dragged around several objects in a drawing. This allows you to quickly drag a selection box around a group of entities and select only certain entities instead of everything inside the box.

Using the Command

Choose the SELECTION FILTER command from the EDIT menu. The Selection Filter dialog box appears. To use a selection filter, click on the checkbox to the left of the entity characteristics to which you want the selection to be restricted. Some of the Selection Filter types allow you to specify characteristics such as color or entity type. As long as the Selection Filter dialog box is open, only those entities you enabled can be selected.

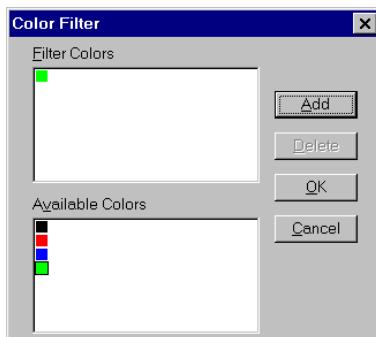
To enable more items for selection, check or select them from the Selection Filter dialog. To disable items, uncheck or remove them from the filter groups in the Selection Filter dialog. To turn off the Selection Filter completely, just close the Selection Filter dialog box by clicking on the x button in the upper-right corner of the dialog box.

To set options for a particular entity's selection, click the appropriate setup button.



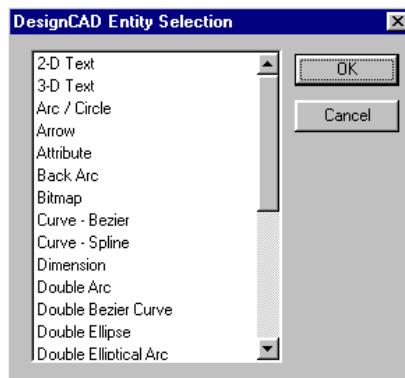
Color Selection Filter

This filter lets you select entities by color. To add a color to the filter so items of that color may be selected, highlight the color in the AVAILABLE COLORS area and click ADD. To remove a color from the filter so items of that color will not be selected, highlight the color and then click DELETE. Click OK to return to the Selection Filter dialog box.



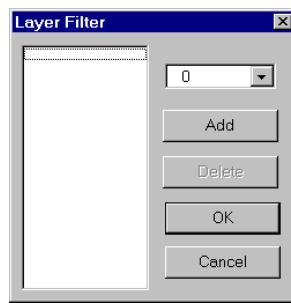
Entity Type Selection Filter

You can pick any entity or combination of entities for a selection box to select. Click to select the entity type you want to select, then click OK.



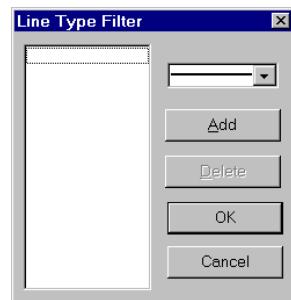
Layer Selection Filter

This filter lets you select entities by layer. To add a layer to the filter so items in that layer may be selected, select the layer number from the list box and click ADD. To remove a layer from the filter so items in that layer will not be selected, highlight the layer number in the area to the left of the list box and then click DELETE. Click OK to return to the Selection Filter dialog box.



Line Type Selection Filter

You can pick any line type or several line types to enable them to be selected. To add a line type to the filter so items of that line type may be selected, select the line type from the list box and click ADD. To remove a line type from the filter so items of that line type will not be selected, highlight the line type in the area to the left of the list box and then click DELETE. Click OK to return to the Selection Filter dialog box.



Group Entity Selection Filter

When this Selection Filter is enabled, only groups may be selected.

Solid Entity Selection Filter

When this Selection Filter is enabled, only solids may be selected.

Selection Zoom Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	ZOOM
Toolbox Icon:	

The Selection Zoom command zooms, or scales, selected entities to be larger or smaller. It is similar to the Scale command, but it changes both the X and Y scales the same amount. This keeps the proportions of the object intact.

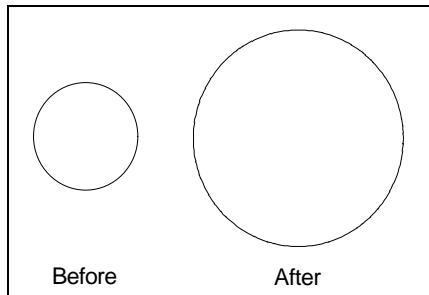
Using the Command

Select the object to be zoomed. Choose ZOOM from the SELECTION EDIT submenu in the EDIT menu. Enter the Zoom Factor in the ZOOM FACTOR box in the dialog box. Press the **Enter** key. The object will be redrawn to the new Zoom Factor.



Example: Zoom a Circle.

Select the circle to be zoomed. Choose ZOOM from the SELECTION EDIT submenu in the EDIT menu. Enter **2** in the ZOOM FACTOR box in the dialog box. Press **Enter**. The circle will be enlarged to twice its original size.



Select Next Command

Menu:	EDIT
Menu Command:	SELECT NEXT

The Select Next command is used to select relatively small entities that are completely overlapped by other entities. If the wrong entity is consistently selected when the mouse is used to click in an area containing a small entity, the Select Next command can be used to select the desired entity.

Using the Command

Attempt to select an entity at a point that is overlapped by another entity. One of the entities is selected and a selection handle appears at the location on which you clicked. Choose SELECT NEXT from the EDIT menu. The other entity that occupies the exact location of the selection handle or a location near the selection handle will be selected.

Example: Select a point mark

Draw a curve. Choose the POINTMARK command from the DRAW menu and draw a point mark at one end of the curve. Attempt to select the point mark. If the curve is selected instead, choose the SELECT NEXT command from the EDIT menu. The point mark will be selected instead of the curve.

Select Previous Command

Menu:	EDIT
Menu Command:	SELECT PREVIOUS
Shortcut Key:	Shift+P

The Select Previous command selects the entities that were previously selected in the drawing.

Using the Command

Choose SELECT PREVIOUS from the EDIT menu. The entities previously selected in the drawing will be selected.

Example: Reselect several entities.

Select several entities and then deselect them by pressing **Esc**. Choose the SELECT PREVIOUS command. The previously selected entities will be re-selected.

Select Scanner Source Command

Menu:	FILE
Submenu:	IMAGE
Menu Command:	SELECT SCANNER SOURCE

The Select Scanner Source command selects the scanner for use with the Scan Image Command.

Using the Command

Choose the SELECT SCANNER SOURCE command from the IMAGE submenu of the FILE menu. Select the correct scanner source for use with the Scan Image command from the dialog box.

See Also: Scan Image Command

Semi Circle Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	SEMI CIRCLE
Shortcut Key:)
Toolbox Icon:	
Point 1:	Beginning of the semicircle
Point 2:	Endpoint of the semicircle

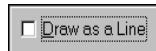
The Semi Circle command draws a semicircle, or 180 degree arc, between two points.

Using the Command

Choose the SEMI CIRCLE command in the Toolbox. Set a point for the beginning of the semicircle. After the first point is set, a rubber-band circle shows how the semicircle will be drawn. Set a point for the end of the semicircle. A semicircle will be drawn counterclockwise from the first point to the second point.

There are two choices for the format of the semi circle, which you can set in the dialog box:

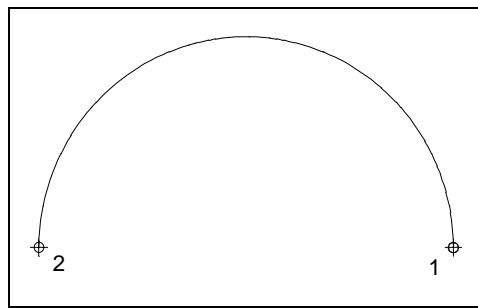
- 1. Arc:** Stored as an actual arc in the drawing (default setting).
- 2. Draw as a Line:** Draws line segments that follow the shape of an semi circle.



The Arc format saves the semi circle as an Arc entity. The Line format saves the semi circle as a series of short line segments. You should normally use the Arc format because it is more efficient and precise. You can use the Line format if you need to distort the semi circle by scaling or stretching.

Example: Draw a semicircle with a diameter of 5 units.

Choose the SEMI CIRCLE command in the Toolbox. Set a point for the beginning of the semicircle. After the first point is set, a rubber-band circle shows how the semicircle will be drawn. Set a point for the end of the semicircle 5 units from the first. A semicircle with a diameter of 5 units will be drawn counterclockwise from the first point to the second point.



Send All Open Documents Command

Menu:	FILE
Submenu:	SEND
Menu Command:	ALL OPEN DOCUMENTS

The Send All Open Documents command is a Windows function that lets you send all the open drawings through Microsoft Exchange for electronic mail and fax functions. For more information, please refer to your Windows documentation.

Send Current Document Command

Menu:	FILE
Submenu:	SEND
Menu Command:	CURRENT DOCUMENT

The Send Current Document command is a Windows function that lets you send the current drawing through Microsoft Exchange for electronic mail and fax functions. For more information, please refer to your Windows documentation.

Set As DesignCAD Tile Command

Menu:	WINDOW
Submenu:	DESIGNCAD TILE SETTINGS
Menu Command:	SET AS DESIGNCAD TILE

The Set As DesignCAD Tile command changes the DesignCAD Tile setting to the current view configuration.

Using the Command

After rearranging the view windows, choose the SET AS DESIGNCAD TILE command from the DESIGNCAD TILE SETTINGS submenu in the WINDOW menu. The DesignCAD Tile setting is set to the current view configuration.

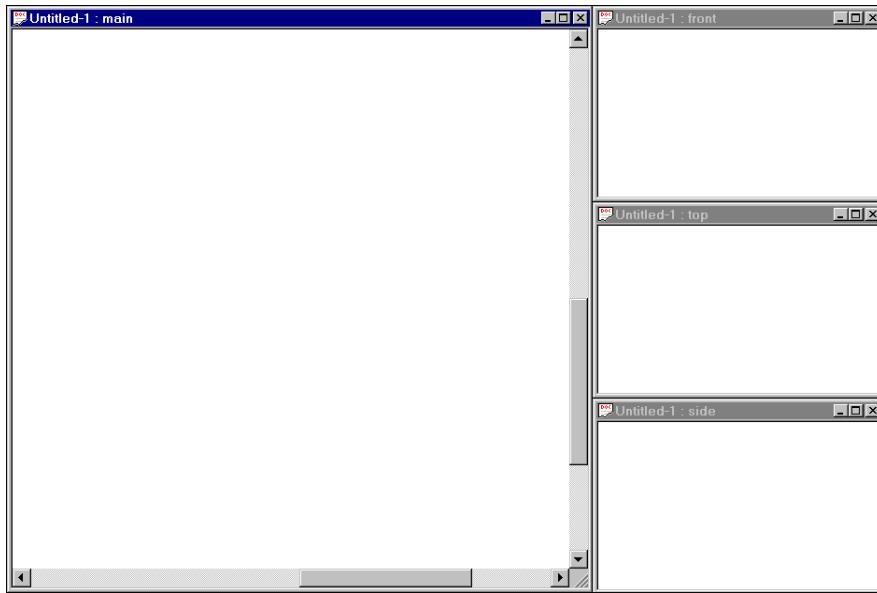
Note: The DesignCAD Tile settings for 2-D Mode and 3-D Mode are stored as different settings; therefore, changes made to the DesignCAD Tile setting while in 2-D Mode will not affect the DesignCAD Tile setting for 3-D Mode, and vice versa.

The Restore DesignCAD Tile command resets the DesignCAD Tile setting to its default view configuration: the Perspective view in the large window on the right side of the screen, and the Front, Top, and Side views stacked vertically on the left.

Example: Reverse the default DesignCAD Tile setting

To move a view window, click in the title bar for the view to be moved. While holding the mouse button down, drag the window to its new position. To "drop" the window, release the mouse button. Move the Front, Top, and Side views from the left side of the screen to the right. Move

the Perspective view from the right side of the screen to the left. Select the SET AS DESIGNCAD TILE command from the DESIGNCAD TILE SETTINGS submenu in the WINDOW menu. Until the DesignCAD Tile setting is changed again using the Set As DesignCAD Tile command or the Restore DesignCAD Tile command, this is the view configuration that will be used every time the DesignCAD Tile command is selected.



See Also: *DesignCAD Tile Command, Restore DesignCAD Tile Command*

Set Drawing Handles Command

Menu:	POINT
Menu Command:	SET DRAWING HANDLES
Point 1:	First handle
Point 2:	Second handle (optional)
Point 3:	Third handle (optional)

The Set Drawing Handles command sets handles for the drawing. Use these handles to help you manipulate a drawing when you merge it with a new or existing drawing using the Load Symbol command.

Using the Command

Select the SET DRAWING HANDLES command from the POINT menu. Set one to three points in the drawing for the handles. If you set less than three points, press **Enter** to end the command.

Next, choose the SAVE AS command in the FILE menu. The SAVE AS dialog box appears. Rename the file or save it with the same name, then click the OK button. If you keep the same name, you are asked if you want to replace the existing file. Click OK.

The drawing is saved with the handles in place. The next time you merge the drawing into another one using the Load Symbol command, you can position the drawing using the handles you have set.

Example: Attach a drawing of a wing to a fuselage in another drawing.

Open the drawing of the wing. Choose the SET DRAWING HANDLES command. Set two points at the base of the wing, where it will connect to the fuselage. Press **Enter**.

Next, choose the SAVE AS command. Rename the file as **SETWING.DCD** and click **OK**. When you merge the wing into the drawing of the fuselage using the Load Symbol command, you can attach the wing precisely using the handles you have set and the Gravity command.

See Also: *Load Symbol Command, Save As Command, Set Handles Command*

Set Grid Center Command

Menu: OPTIONS

Menu Command: SET GRID CENTER

Point 1: Set a point for the display grid center

The Set Grid Center command sets the location for the center of the display grid.

Using the Command

Choose the SET GRID CENTER command. Set a point anywhere on the drawing screen to specify the new center of the grid.

The display grid can be oriented on the X-Y, X-Z, or Y-Z plane. Its size, orientation, and spacing can be set in the GRID OPTIONS folder.

See Also: *Grid Options*

Set Handles Command

Menu: EDIT

Submenu: SELECTION EDIT

Menu Command: SET HANDLES

Shortcut Key: **Ctrl+H**

Point 1: First handle

Point 2: Second handle (optional)

Point 3: Third handle (optional)

The Set Handles command can be used to set selection handles. Selection Handles are reference points for one or more entities that have been selected. The selection handles are used to help you move, copy, or manipulate the selection.

Using the Command

Select the object on which you want to set the handles. Choose the SET HANDLES command. Set from one to three handles on the object. The first handle set is the primary handle and is designated by a bull's eye. The secondary handle is represented by a small 1 enclosed in a circle.

The tertiary handle is a small 2 enclosed in a circle. Press Enter to end the command if you set one or two handles. Now you can manipulate the object using the handles.

For example, when you copy an object with the Duplicate command, the first selection handle is located at the destination point you set. If you place the selection handle at a convenient location, it can be much easier to position the copy.

The second selection handle is used with some commands to set the size and angle of a selection. The Duplicate and Move commands use the second handle.

The third handle is sometimes used to position the selection in 3-D space. For example, the Move command locates the first two handles on the first two points set, and then rotates the selection so that the three handles lie on the same plane as the three points set.

Example: Set handles on a box and move it.

Select the box. Choose the SET HANDLES command. Set the first handle on the upper-right corner, the second on the lower-left corner, and the third on the lower-left-rear corner. Next, choose the MOVE command. Move the cursor to the desired new location for the box and set a point for the first handle. Move the cursor away from the first point. A rubber-band scaling box is drawn using the cursor location as Point 2. When you have scaled the object to your liking, set the second handle. Set a point for the third handle to orient the object in 3-D space.

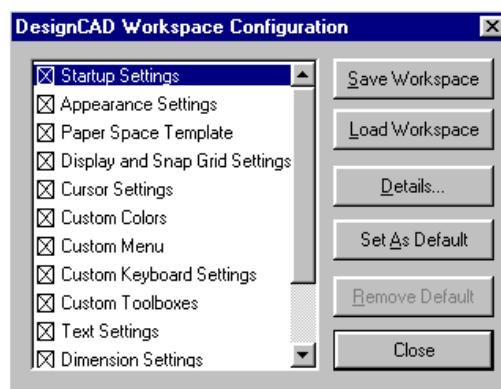
Setup DesignCAD Workspace Command

Menu:	FILE
Submenu:	WORKSPACE CONFIGURATION
Menu Command:	SETUP DESIGNCAD WORKSPACE

The Setup DesignCAD Workspace command is used to configure various sets of DesignCAD settings so that they can be saved to disk and then retrieved in the future. This feature lets you simply open a set of options instead of having to go through and set many options at the beginning of every drawing session.

Using the Command

Select the SETUP DESIGNCAD WORKSPACE command from the WORKSPACE CONFIGURATION submenu in the FILE menu. The DesignCAD Workspace Configuration dialog box appears.



To save a group of settings in the current Workspace template, place an x in the box to the left of the group name by clicking on it with the mouse. To remove a group of settings from the current Workspace template, click on the box to remove the x in the box to the left of the group name. If you want to change a group of settings, click on the group name to highlight it and then click on the DETAILS button.

Save Workspace

Click this button to save the settings as a Workspace. The Save DesignCAD Workspace dialog box opens. Enter a name for the file and where you want to save it. Then click OK.

Load Workspace

To base a new Workspace on an existing one, click on the LOAD WORKSPACE button and specify the workspace on which the new one will be based.

Details

If you want to change a group of settings, click on the group name to highlight it and then click on the DETAILS button.

Startup Settings

This group of settings controls the startup settings for DesignCAD drawing sessions. These settings include:

- Startup macros that have been recorded or written
- Paper Size and Margins
- Whether DesignCAD opens in 2D Mode or 3D Mode
- Units of Measurement for DesignCAD drawings
- Number of Units across the Main View
- Left-hand or Right-hand Coordinate System

Appearance

This group of settings controls the colors of various DesignCAD elements. Simply click on the color sample to the right of the name of the element for which the color is to be changed and select the new color from the palette that appears. The elements for which the color can be changed include:

- 2D Cursor
- Individual axes of the 3D Cursor
- Background
- Grid
- Rubber Band
- Point
- Selection
- Selection Handle
- Selection Box

- Resizing Handles

Paper Space Template

Select the Paper Space Template to be used with the current DesignCAD Workspace.

Display Grid / Snap Grid Settings

This group of settings determines the settings for the Display and Snap Grids that will be used when the current DesignCAD Workspace is selected. The Display Grid and Snap Grid settings which can be changed include:

- Snap Grid on/off
- Snap Grid size
- Display Grid on/off
- Display Grid size
- Display Grid extent
- Display Grid plane
- Display Grid type

Cursor Settings

This group of settings sets the size of the cursor and the small and large step sizes that will be used in the current DesignCAD Workspace.

- Cursor Step sizes
- Fixed Cursor Size

Custom Colors

This group of settings determines the custom colors that will be available in the drawings that use the current DesignCAD Workspace.

Custom Menu

Select the Customized Menu file to be used with the current DesignCAD Workspace by clicking on the BROWSE button.

Custom Keyboard

Select the Customized Keyboard file to be used with the current DesignCAD Workspace by clicking on the BROWSE button.

Custom Toolboxes

Select the Customized Toolboxes file to be used with the current DesignCAD Workspace by clicking on the BROWSE button.

Text Settings

This group of settings sets the default color and style of regular text and Attribute text for the different layers of drawings that use the current DesignCAD Workspace.

Dimension Settings

This group of settings determines the options for the dimensioning commands, Pullout command, and Balloon commands that will be used when the current DesignCAD Workspace is specified.

Pointmark Settings

This group of settings determines the default options for pointmarks. These settings will be used when the current DesignCAD Workspace is specified. Pointmark Settings include:

- Layer in which Pointmarks will be drawn
- Pointmark color
- Pointmark type
- Pointmark size

Hatch Settings

This group of settings determines the default options for line hatching. These settings will be used when the current DesignCAD Workspace is specified. Hatch Settings include:

- Layer in which line hatching will be drawn
- Hatch color
- Hatch Scale
- Hatch Angle
- Hatch Style

Set as Default

Click this button to set the current workspace template so it will be opened automatically at the beginning of every drawing session.

Remove Default

Click this button revert to the original DesignCAD settings.

Close

Clicking on this button closes the dialog box and returns you to your drawing.

See Also: *2-D Mode Command, Color Options, Paper Space Configuration Command, Paper Space Mode Command, Grid Options, Cursor Options, Custom Color Command, Menu Options, Keyboard Options, Toolbox Options, Text Options, Dimension Options, Point Mark Command, Hatch Command*

Set View Command

Menu:	VIEW
Menu Command:	SET VIEW
Shortcut Key:	Y
Toolbox Icon:	

The Set View command is used to change the viewer location.

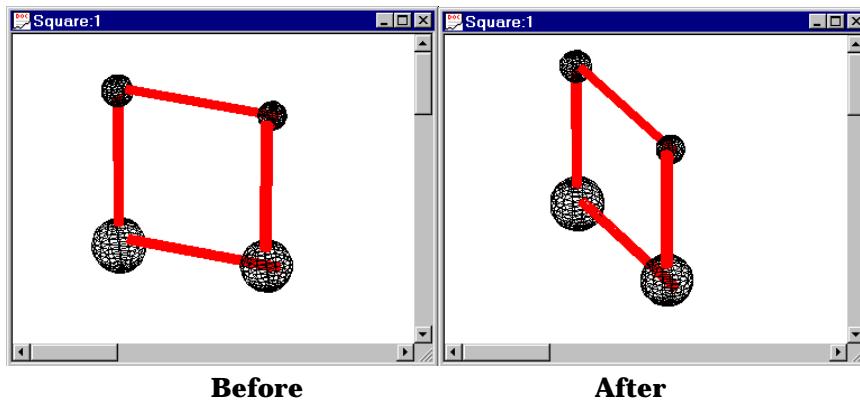
Using the Command

Select the SET VIEW command and move the cursor to the view window. Drag the cursor across the screen with the mouse. The view changes as you drag the mouse. To preview the drawing at a view position, release the mouse button. Press **Enter** when you have the desired view.

You can change the perspective of the view by moving the cursor in and out on the Z axis (by pressing **Ctrl+Shift** and moving the mouse). The Set View command can be undone with the Zoom Previous command.

Example: See how your drawing looks from several different view angles.

Select the SET VIEW command. The cursor turns into a camera. Hold down the left mouse button while moving the mouse to set the view position. To see how the drawing looks in the new view, let go of the mouse button. If the view needs further adjustment, hold down the left mouse button and move the mouse again. When the view is set to your liking, click **OK** in the Command Line. To return to the original view, click **CANCEL**.



Set Working Plane Command

Menu:	VIEW
Submenu:	WORKING PLANE
Menu Command:	SET WORKING PLANE
Point 1:	Origin of workplane
Point 2:	X axis of workplane
Point 3:	Orientation of workplane
Point 4:	A point defining the front of the plane

The Set Working Plane command can be used to choose a specified plane as a temporary substitute for the XY plane. It adjusts mouse and cursor-key movements accordingly. This feature is very convenient for doing work along oblique faces of complex objects.

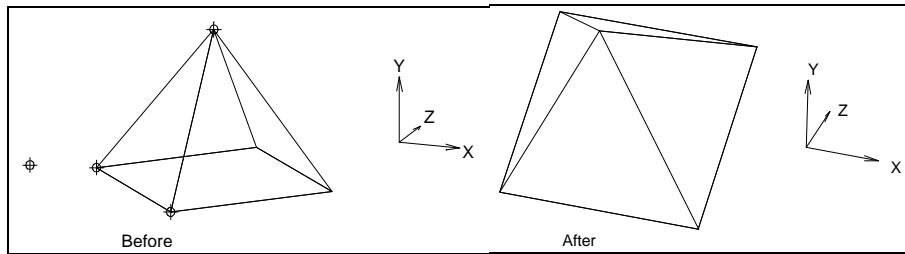
Using the Command

Choose the SET WORKING PLANE command. Set four points for the workplane. Point 1 becomes the new origin. Point 2 determines the X axis of the workplane. Point 3 establishes the orientation of the plane. Point 4 determines the front side of the plane.

Note: If the entire drawing is not visible after using the Set Working Plane command, use the Fit to All Windows command to readjust the views.

Example: Work on the face of a pyramid.

Select the SET WORKING PLANE command. Set the first point on the lower-left corner of one of the faces. This point sets the origin for the workplane. Set the second point at the lower-right corner of the same face. This sets the X axis for the workplane. Set the third point on the apex of the pyramid. This sets the orientation, or tilt, of the plane. The last point determines which side of the plane DesignCAD recognizes as the front. When you have done this, the program redraws the pyramid with the front of the plane facing you.



See Also: [Reset Working Plane Command](#)

Shading Command

Menu: **TOOLS**

Menu Command: **SHADING**

Shortcut Key: **F8**



Toolbox Icon:

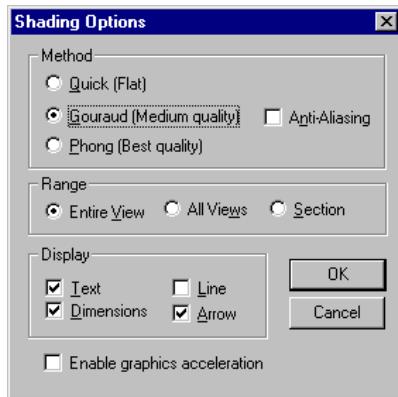
Point 1: First corner of shading region (optional)

Point 2: Opposite corner of shading region (optional)

The Shading command shades all or part of a drawing. Only solids and planes can be shaded.

Using the Command

Choose the SHADING command. In the Shading Options box determine the Method and range of shading you want, whether you want to display text, dimensions, lines, or arrows when the drawing is finished shading, and whether you want to use the Anti-Aliasing option.



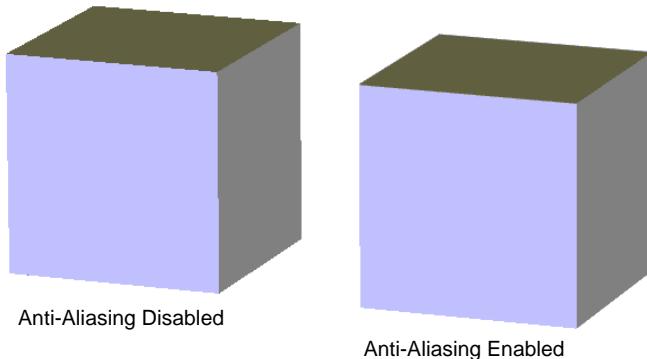
Phong (Best Quality) shading uses a fine shading pattern, and displays the material texture and/or texture mapping.

Gouraud (Medium Quality) shading displays the material texture and/or texture mapping, but the resulting shading is not quite as fine as that of Phong shading.

Quick (Flat) shading is quite a bit faster than Phong or Gouraud shading, but the shading is coarser and less realistic. No smoothing takes place, and material textures and texture mapping are not represented.

All three shading methods can be done either for the entire screen or for a rectangular region of the screen. The method you choose becomes the default selection the next time you use the Shading command.

The Anti-Aliasing option tells DesignCAD whether or not to change the color of the object at its edges slightly so that all straight edges will appear smoother. If the Anti-Aliasing option is checked the edges will be smoothed. If the Anti-Aliasing option is not checked some of the edges will appear slightly jagged.



If your graphics card supports graphics acceleration, check the ENABLE GRAPHICS ACCELERATION option to improve performance. This will enable interactive shading. Interactive shading will allow you to move and edit existing drawing items and even add new ones, all while the specified view(s) maintain shading. The view(s) will remain in shading mode until you change the status of the view(s) to wireframe or hidden line removal.

If Windows is configured for only 16 colors, the shaded drawing is not very realistic. If Windows is configured for 32,000 or more colors, however, the shading will be fastest and look very realistic.

You can change the number of colors and the screen resolution used by Windows in the Windows Control Panel, under Display.

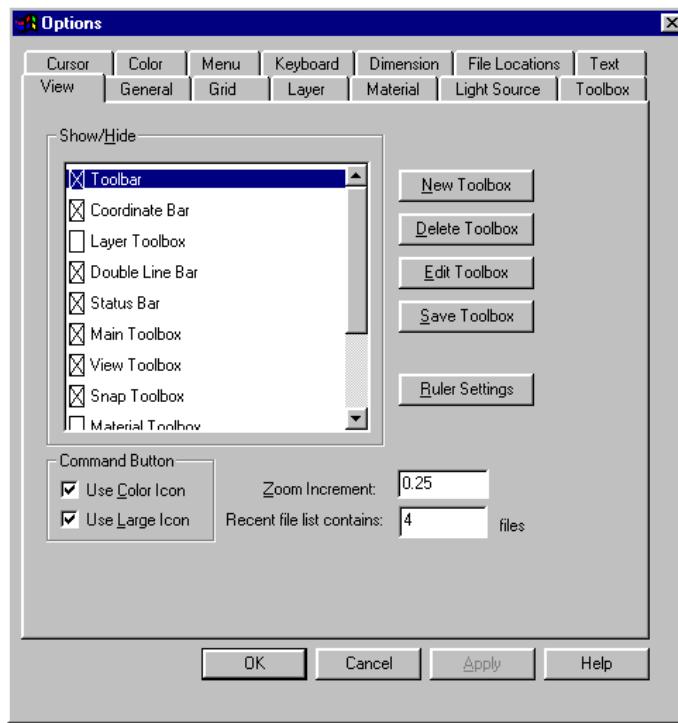
Show/Hide Command

Menu: **VIEW**
Menu Command: **SHOW/HIDE**

The Show/Hide command brings up the View Options folder and lets you select the toolboxes and bars you want to display on the screen.

Using the Command

Choose the SHOW/HIDE command, which brings up the VIEW OPTIONS folder. Click on the checkboxes to select the items that you want to be visible on the screen. A marked box indicates an item will be shown on the screen. Click OK.



Sketch Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	SKETCH
Toolbox Icon:	

The Sketch command draws freehand by following the cursor's movement.

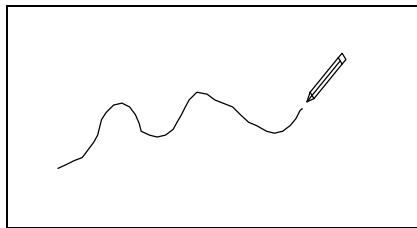
Using the Command

Choose the SKETCH command in the Toolbox. Press and hold down the left mouse button. Move the cursor in a curve motion. Release the mouse button. A curve is drawn, following the cursor's movement.

Note: The Sketch Command only works on the current workplane in 3-D Mode. The default workplane is the XY plane. This plane can be changed in the General Options folder of the Options file box. See the General Options entry in the "Command Reference" section.

Example: Draw a freehand curve.

Choose the SKETCH command in the Toolbox. Press and hold down the left mouse button. Draw a curve on the screen. Release the mouse button. The curve will follow the exact path of the cursor.



Note: The Smooth Line by Point Reduction and Smooth Line by Slope Detection commands can be used to "clean up" a line drawn with the Sketch command.

See Also: *Smooth Line by Point Reduction Command, Smooth Line by Slope Detection Command*

Skew Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	SKEW
Toolbox Icon:	
Point 1:	Reference point
Point 2:	New position for reference point

The Skew command slants one or more selected entities along the X, Y, Z or "free" axis.

Using the Command

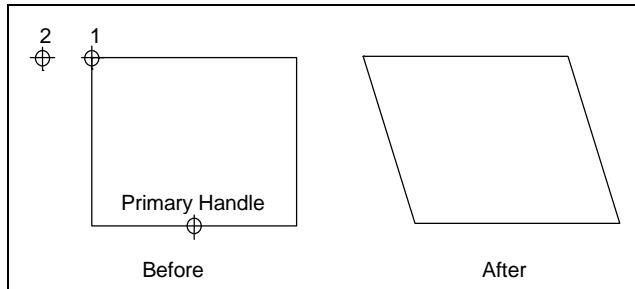
Select the entities to be skewed. Choose the SKEW command from the SELECTION EDIT submenu of the EDIT menu. In the dialog box, choose the axis on which the selected objects are to be skewed. Set a reference point in the drawing. A rubber-band representation of the entity being skewed shows how the entity will be drawn. Set a second point away from the reference point at the same distance and in the direction that the objects should be skewed. The objects are skewed according to the distance and direction specified.



Note: The Primary Selection Handle of the selected object serves as an anchor during the Skew Command. If this handle is located in the center of the object, both sides of the object move from their original positions.

Example: Skew a box.

Select the box. Set the Primary Selection Handle on the center of the bottom side of the box so it will not be moved. Choose the SKEW command. Select the X axis in the dialog box. Set a reference point at the top-left corner of the box. A rubber-band box is drawn to show how the box will be drawn. Set a second point 5 Drawing Units to the left of the reference point. The box is skewed so the top-left corner of the box is 5 Drawing Units to the left of its original position.



Slice Command

Menu: SOLIDS

Menu Command: SLICE

Shortcut Key: **Ctrl+L**

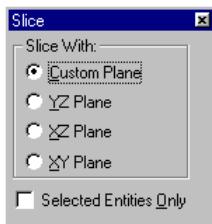
Points 1-3: Points to define the cutting plane

Point 4: Region to be removed

The Slice command is used to erase parts of the drawing in front of or behind a predefined or custom plane.

Using the Command

Select the SLICE command. The Slice dialog box appears.



If the CUSTOM PLANE option is selected, set three points to define the "cutting plane." Then set a point either in front of or behind the cutting plane. All parts of the drawing that lie on that side of the cutting plane will be erased.

If the fourth point is not used and the **Enter** key is pressed, then all entities and parts of entities will be cut at the cutting plane location, but will not be erased.

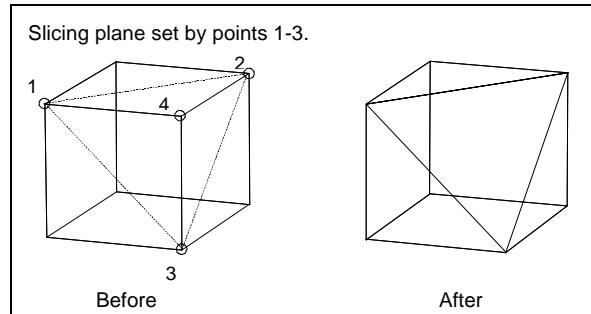
If the YZ PLANE, XZ PLANE, or XY PLANE option is selected, set a single point to define the location of the "cutting plane." Then set a point either in front of or behind the location of the cutting plane. All parts of the drawing that lie on that side of the cutting plane will be cut and erased at the specified location using a plane that lies parallel to the plane indicated in the selected option.

If the second point is not used and the **Enter** key is pressed, then all entities and parts of entities will be cut at the cutting plane location, but will not be erased.

You can choose to run the Slice command on the current selection only by activating that option in the dialog box.

Example: Cut a selected box along a plane.

Select the SLICE command. Click the CUSTOM PLANE option and the SELECTED ITEMS ONLY checkbox. Set the first point on any corner of the box. Set the second point on the corner that is diagonally across the same face of the box. Set a third point on the corner that is diagonally across a face from Point 2. Set the last point on the side of the box that you want removed. The box will be redrawn with the selected portion sliced away.



Slice by Curved Surface Command

Menu: SOLIDS

Menu Command: SLICE BY CURVED SURFACE

Point 1: Solid to be sliced

Point 2: Surface to be used to slice solid

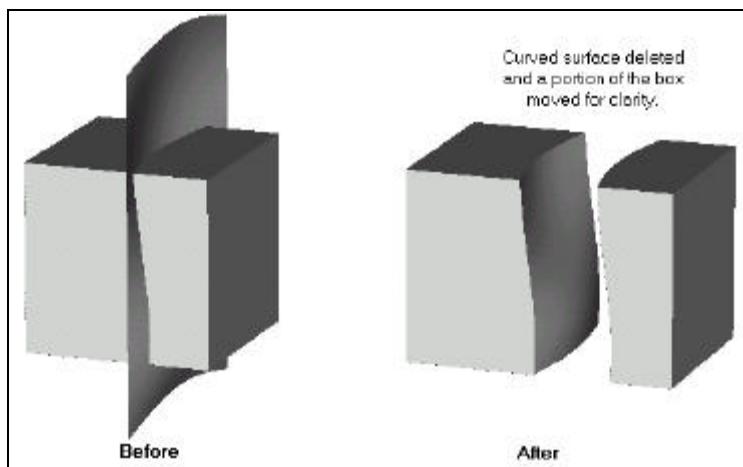
The Slice by Curved Surface command is used to slice a solid using a curved surface.

Using the Command

Select the SLICE BY CURVED SURFACE command. Set a point on the solid to be sliced. Set a point on a curved surface that intersects the solid. The curved surface may have been drawn using the Surface Patch or Surface Connect command. The solid is cut using the curved surface.

Example: Cut a box using a curved surface.

Select the SLICE BY CURVED SURFACE command. Set the first point anywhere on the box. Set the second point anywhere on the curved surface. The box is cut using the curved surface.



See Also: Slice Command

Smooth Line by Point Reduction Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	SMOOTH LINE BY POINT REDUCTION

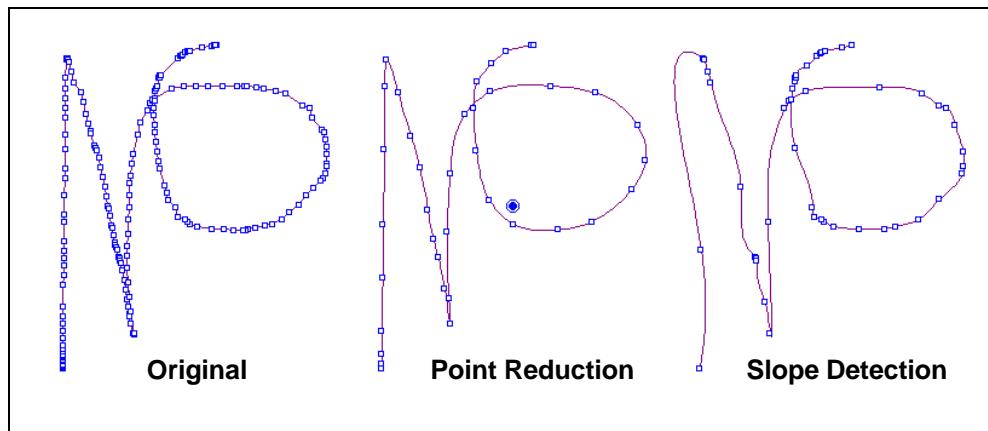
The Smooth Line by Point Reduction command smoothes complex lines and curves by reducing the number of points in them. DesignCAD divides the points of the line(s) and/or curve(s) into groups of five. The locations of the points in each group are averaged, and the group is replaced with a single point. Lines will be automatically converted to curves by this command.

Using the Command

Select the lines and/or curves to be smoothed. Run the SMOOTH LINE BY POINT REDUCTION command from the SELECTION EDIT submenu of the EDIT menu. The objects are smoothed.

Example: Smooth a complex line.

Select the line. Choose the SMOOTH LINE BY POINT REDUCTION command. The line is converted to a smooth curve. The figure below is shown with Point Select Mode turned on to illustrate the reduced number of points in the result. The original line is shown along with the results of a Smooth Line by Point Reduction and a Smooth Line by Slope Detection.



See Also: [Line to Curve Command](#), [Curve to Line Command](#), [Smooth Line by Slope Detection Command](#), [Vector Convert Command](#)

Smooth Line by Slope Detection Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	SMOOTH LINE BY SLOPE DETECTION

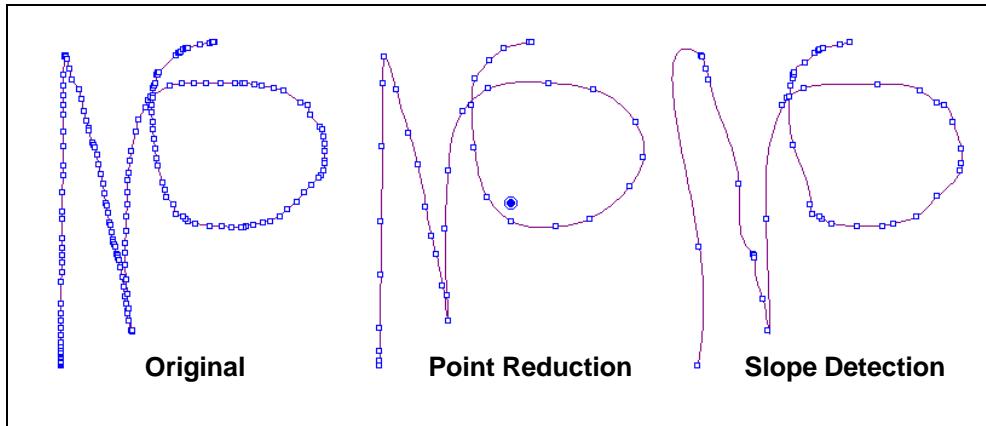
The Smooth Line by Slope Detection command smoothes complex lines and curves. A point is removed if the two segments that it joins do not deviate significantly from a straight line. Lines will be automatically converted to curves by this command.

Using the Command

Select the lines and/or curves to be smoothed. Run the SMOOTH LINE BY SLOPE DETECTION command from the SELECTION EDIT submenu of the EDIT menu. The objects are smoothed.

Example: Smooth a complex line.

Select the line. Choose the SMOOTH LINE BY SLOPE DETECTION command. The line is converted to a smooth curve. The figure below is shown with Point Select Mode turned on to illustrate the reduced number of points in the result. The original line is shown along with the results of a Smooth Line by Point Reduction and a Smooth Line by Slope Detection.



See Also: [Line to Curve Command](#), [Curve to Line Command](#), [Smooth Line by Point Reduction Command](#), [Vector Convert Command](#)

Snap Grid Command

Menu:	OPTIONS
Menu Command:	SNAP GRID
Shortcut Key:	G

The Snap Grid command toggles the snap grid off and on. With Snap Grid enabled, any time you set a point with the mouse, the point is set at the nearest location on the grid.

See Also: [Grid Settings Command](#)

Solid Add Command

Menu:	SOLIDS
Menu Command:	SOLID ADD
Shortcut Key:	Ctrl+J
Point 1:	First Solid
Point 2:	Solid to be added to first

The Solid Add command is used to "add" one Solid object to another. It makes a single Solid out of two Solids and eliminates the unnecessary surfaces after the addition of the two Solids.

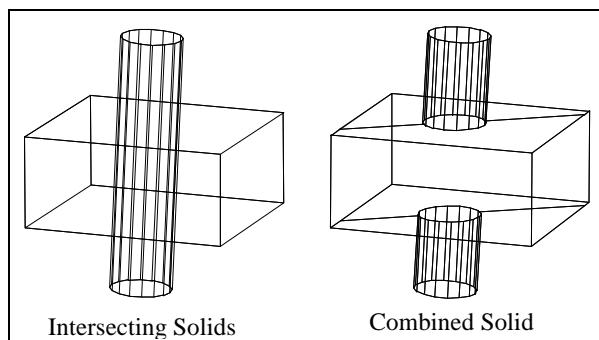
Using the Command

Choose the Solid Add command. Set a point on each of the Solids to be added. The Solids are then combined into a single Solid, and the surfaces "internal" to the resulting Solid are removed.

Note: For best results, the solids should overlap rather than meet exactly at a single face. For example, if you want to stack a cube on top of another, make one cube just a little taller, and place the other cube so that the overall height is correct. When you perform the Solid add, the intervening volume is removed.

Example: Make a single solid from a cylinder passing through a box.

Select the SOLID ADD command and set a point on the box. Set a second point on the cylinder. DesignCAD redraws the two objects as a single Solid, removing the shared volume.



Solid Define Command

Menu:	SOLID
Menu Command:	SOLID DEFINE
Shortcut Key:	Ctrl+D

The Solid Define command makes DesignCAD recognize all the currently selected entities as a single Solid. This allows you to build Solids from a collection of planes, extrusions, and surfaces.

Using the Command

Select the objects you want to define as one Solid. Choose the SOLID DEFINE command. The Solids are defined as one Solid.

A Solid is a set of planes and surfaces that make up a closed Solid object. A Solid must be defined before performing Solid operations such as Solid Subtract, Solid Add, Solid Intersect, and Interference Checking.

A Solid must be a closed set of planes and surfaces. The Solid Define command does not verify whether the current selection is a legitimate Solid. If you define an invalid Solid, operations such as the Solid Subtract command may not work correctly.

See Also: **Solid Add Command**, **Solid Explode Command**, **Solid Subtract Command**

Solid Explode Command

Menu:	SOLID
Menu Command:	SOLID EXPLODE

The Solid Explode command can be used to explode a Solid. It reduces a selected Solid into its component parts: planes, surface meshes, and lines. Each component can then be manipulated individually, without affecting the rest of the former Solid.

Using the Command

Select the Solid to be exploded. Choose the Solid Explode command. The Solids now regain their individual properties and can be manipulated as individual Solids.

A Solid is a set of planes and surfaces that make up a closed Solid object. A Solid must be defined before performing Solid operations such as Solid Subtract, Solid Add, and Interference Checking.

Solid Intersect Command

Menu:	SOLID
Menu Command:	SOLID INTERSECT
Point 1:	First Solid
Point 2:	Second Solid

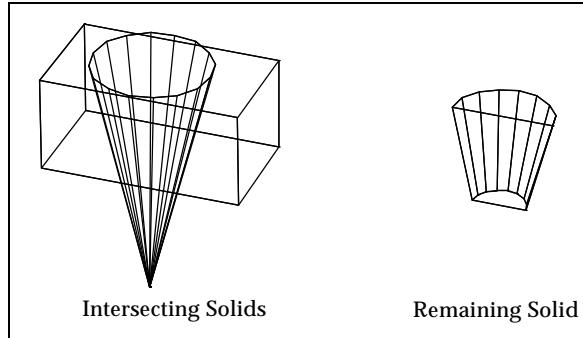
The Solid Intersect command removes all parts of two overlapping Solids except the part that both Solids share. This part forms a new Solid.

Using the Command

Choose the SOLID INTERSECT command. Set a point on each of the intersection Solids. The overlapping area remains.

Example: Create a new Solid out of the volume shared by two intersecting Solids.

Select the SOLID INTERSECT command. Set a point on the first Solid, then a point on the second Solid. DesignCAD redraws the new object as a Solid formed by the area previously shared by both objects. The parts of the Solids which did not intersect are removed.



Solid Subtract Command

Menu: SOLIDS
 Menu Command: SOLID SUBTRACT
 Shortcut Key: **Ctrl+U**
 Point 1: Solid to be removed (template)
 Point 2: Solid to be subtracted from

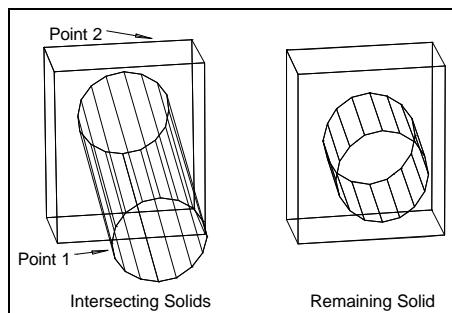
The Solid Subtract command removes one Solid from another Solid that it overlaps. For example, to drill a round hole in a Solid object, you can subtract a cylinder from it.

Using the Command

Draw two solids that intersect one another. Choose the SOLID SUBTRACT command. Now set a point anywhere on the solid to be subtracted (the one that will be used as a template for the hole). Set a second point on the solid to be subtracted from. The solid specified with the first point is removed from the solid specified by the second point, leaving all the solid except the part that coincided with the template.

Example: Cut a cylinder out of a box that it intersects.

Select the SOLID SUBTRACT command and set a point on the cylinder. Now set a second point on the box. The resulting Solid is a box with a hole through it.



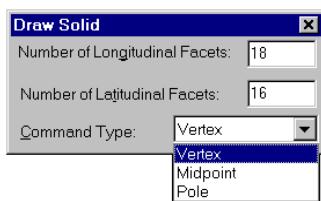
Sphere Command

Menu: SOLIDS
 Menu Command: SPHERE
 Toolbox Icon: 
 Point 1: Center of the sphere
 Point 2: Radius of the sphere

The Sphere command draws a solid sphere.

Using the Command

Choose the SPHERE command. You can specify the number of sides or facets around both the longitude and latitude of the sphere in the NUMBER OF LONGITUDE AND LATITUDE boxes in the dialog box. The more facets the sphere has, the more spherical it appears.



The command requires two points to be set: one for the center of the sphere and one for the radius. You can choose whether the radius of the sphere will be set at a vertex of the equator, a midpoint of the equator, or at one of the sphere's poles. If you choose VERTEX, the equator of the sphere is inscribed by a circle of that radius. If you choose MIDPOINT, the equator of the sphere circumscribes a circle of that radius. If you choose POLE, the second point will determine one of the two poles for the sphere. This is normally not significant, but it can be important for some precision drawings.

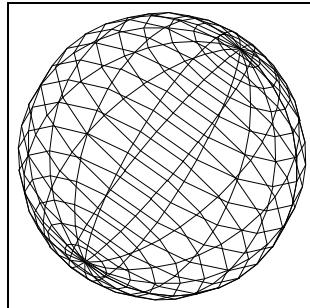
It may help to think of the sphere as planet Earth. The first point is the center of the Earth. The second point determines the radius of the planet. This point can lie in one of three places: Pole, Vertex, or Midpoint.

The first point is always at the center of the sphere. If you choose POLE, the second point is at the pole and the axis lies along the line between Points 1 and 2. If you choose VERTEX, the second point lies on the equator at one of the longitudinal divisions. If you choose MIDPOINT, the second point will be on the equator midway between two longitudinal lines.

The number of facets represents how many longitude sections (divisions along the equator) and the number of latitude sections (divisions from the North pole to the South pole).

Example: Draw a sphere.

Select the SPHERE command. Set a point for the center. When you move the cursor away from Point 1, a rubber-band sphere appears. It uses the cursor location as Point 2. When the radius of the sphere and the axis of the poles in the X-Y plane are to your liking, set the second point. The sphere is inserted into your drawing.



Stop Macro Command

Menu:	TOOLS
Menu Command:	STOP MACRO
Shortcut Key:	&

The Stop Macro command halts the execution of a DesignCAD macro. However, you cannot access the Command Menu or the shortcut key to stop the macro if a drawing or shading or similar command is in operation.

Stop Recording Command

Menu:	TOOLS
Menu Command:	STOP RECORDING

The Stop Recording command ends the Macro Record command.

Using the Command

After you have finished recording a macro, choose STOP RECORDING from the TOOLS menu. The macro will be saved under the file name entered in the Macro Record command.

See Also: *Macro Record Command, Record Options Command*

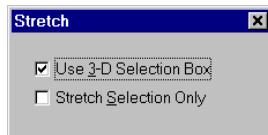
Stretch Command

Menu:	EDIT
Submenu:	SECTION EDIT
Menu Command:	STRETCH
Point 1:	1st corner of area to be stretched
Point 2:	2nd corner of area to be stretched
Point 3:	Reference point
Point 4:	Stretched position of reference point

The Section Stretch command is used to stretch a section of a drawing from one location to another.

Using the Command

Choose the STRETCH command. You can choose to use a 2-D selection box or a 3-D selection box in the dialog box. The 3-D selection box allows you to be more specific when you stretch 3-D objects. For instance, you can use a 3-D selection box to specify just the front left corner of a box.

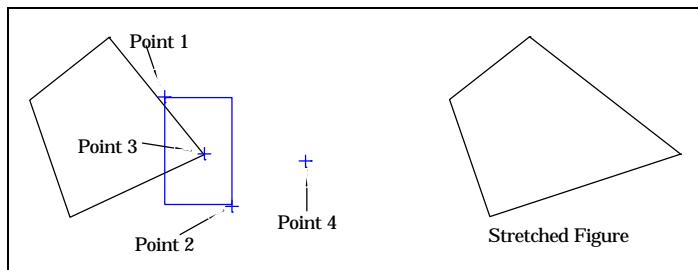


Four points are used with the command. The first two points define a two-dimensional or three-dimensional bounding box that contains the part of the drawing to be stretched. The third and fourth points determine the direction and distance that part of the drawing will be stretched.

For instance, if you move the fourth point 10 Drawing Units along the X axis from the third point, then all points in the stretch region will be moved 10 Drawing Units along the X axis.

Example: Stretch a portion of a plane.

Select the STRETCH command. Make sure the USE 3-D SELECTION BOX option is unchecked. Set Points 1 and 2 for the bounding box so that a corner of the plane is enclosed. Set Point 3 on the corner point of the plane. Set the final point as the new location for Point 3. The plane is redrawn with the section stretched to the new point.



Surface Area Command

Menu: DIMENSION

Submenu: INFO

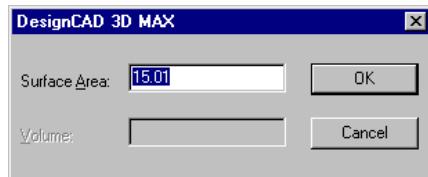
Menu Command: SURFACE AREA

Point 1: Object for which surface area is to be calculated

The Surface Area command calculates the surface area of an object.

Using the Command

Choose the SURFACE AREA command and set a point on the object for which you want DesignCAD to calculate the surface area. The calculation is displayed in the following box:



Example: Calculate the area of a surface in the drawing.

Select the SURFACE AREA command and set a point on the surface. Its surface area is displayed on the screen.

See Also: *Volume Command*

Surface Connect Command

Menu: DRAW
 Menu Command: SURFACE CONNECT
 Shortcut Key: **Ctrl+Q**

Toolbox Icon:

Point 1: First line to be connected with a surface

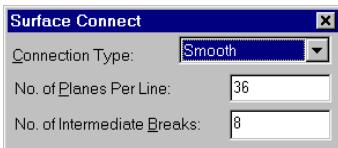
Intermediate Points: Lines or points to define surface path

Point n: Last line to be connected with a surface

The Surface Connect command is used to stretch a surface between two or more lines. The lines can be planes, lines, curves, arcs, or circles. The original lines to be connected can be any shape and at any orientation in 3-D space.

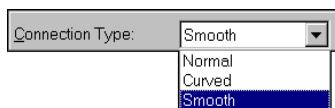
Using the Command

Choose the SURFACE CONNECT command. Set a point on each line entity to be connected and changed into a Solid. Press **Enter** to end the command, when you have the necessary points.



Connecting Surface

There are three options for the connecting surface:



Normal

Makes a ruled surface between each pair of lines.

Curved

Forms a ruled surface "bent" to follow the curve defined by the points set between the original lines. It is a straight or linear fit between the original lines.

Smooth

Forms the smoothest possible surface over the original lines.

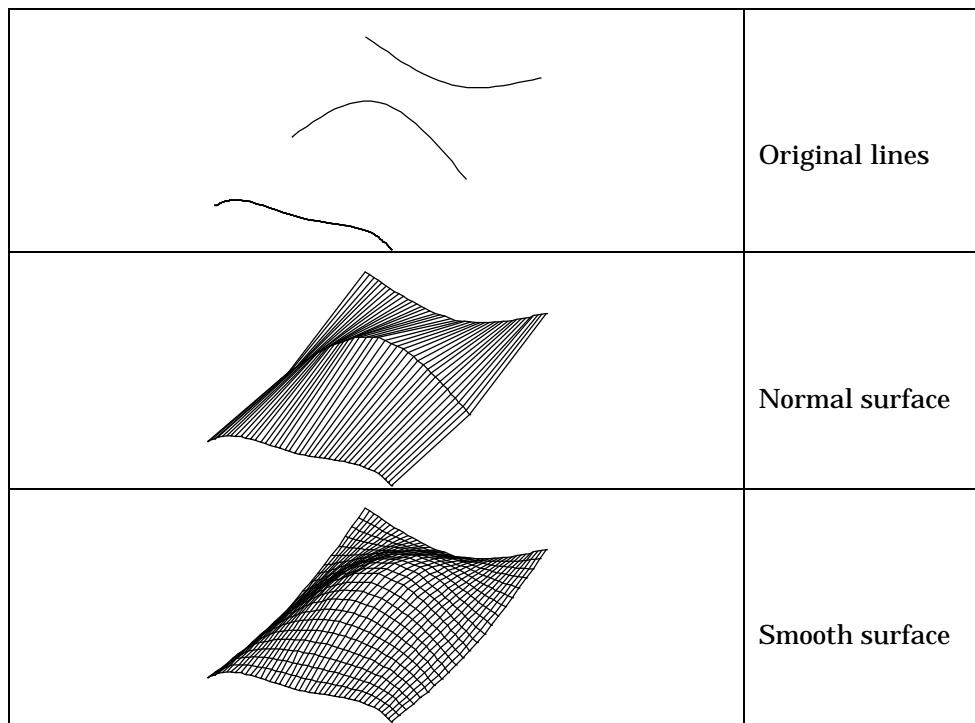
Planes

This option refers to the number of individual planes to be constructed in the connecting surface *along* the original lines.

Intermediate Breaks

This option refers to the number of planes to be constructed in the connecting surface *between* each pair of original lines.

With the Normal and Smooth options, you set a point on each of the original lines to be connected. With the Curved option, you can set additional points in between the original lines to force the surface to curve through those intermediate points.

**Example: Connect three lines with a smooth surface.**

Select the SURFACE CONNECT command. Select the SMOOTH option, 40 planes, and 8 INTERMEDIATE BREAKS. Then set a point on each of the three lines and press **Enter**. The lines are connected by a surface.

Surface Intersection Command

Menu: DRAW
 Submenu: LINES
 Menu Command: SURFACE INTERSECTION
 Toolbox Icon: 
 Point 1: Point on first surface or plane
 Point 2: Point on second surface or plane

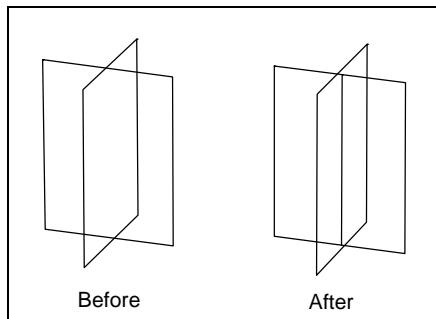
The Surface Intersection command draws a line at the intersection of two surfaces. This command can be used on grid surfaces as well as planes.

Using the Command

Choose the SURFACE INTERSECTION command. Set a point on each of the intersecting surfaces. A line is drawn along the intersection.

Example: Draw a line at the intersection of a curved surface and a plane.

Select the SURFACE INTERSECTION command. Set a point on the curved surface and a point on the plane. A line is drawn along the intersection.



Surface Patch Command

Menu: DRAW
 Menu Command: SURFACE PATCH
 Toolbox Icon: 
 Point 1: First line
 Point 2: Second line
 Point 3: Third line
 Point 4: Fourth line (optional)

The Surface Patch command fits a smooth surface between any three or four lines, curves, or arcs which meet at their endpoints to form a closed area.

Using the Command

Choose the SURFACE PATCH command. Set a point on each line entity you want to patch. Press Enter when you are finished. The lines are then connected with a grid surface.

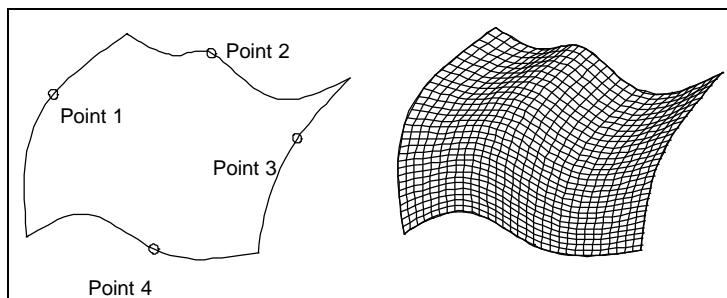
You can specify the grid spacing for the surface:



Note: The line entities must share endpoints and make up a closed area or the Surface Patch command does not work properly.

Example: Create a grid surface out of four lines or curves that meet at their endpoints to form a closed object.

Select the SURFACE PATCH command. Set a point on each of the four lines. When you are finished, press **Enter**. The lines are connected with a grid surface.



See Also: *Surface Connect Command*

Sweep Command

Menu: DRAW

Menu Command: SWEEP

Shortcut Key: W



Toolbox Icon:

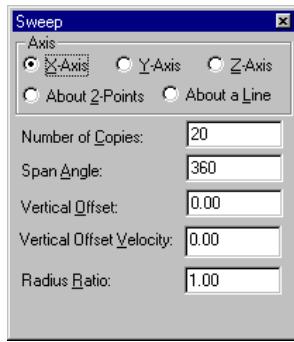
Point 1: Center of rotation

Point 2: Second point for axis of rotation (Two Point rotation only)

The Sweep command makes a circular extrusion of selected objects. This command is used to create a circular Solid object from a single line outlining the silhouette of the object. You can also use this command to make a spiral-shaped object such as a spring or a bolt by specifying an offset for the sweep.

Using the Command

Select the object you want to sweep, and choose the SWEEP command.



Axis

You can select from five options for the axis of rotation. The object is swept around the first point set, parallel to the axis specified here.

X-, Y-, and Z-Axes

Each of these options sweep the object around the selected axis centered at the first point.

About Two Points

This option can be used to define the axis with two points.

About a Line

The LINE option can be used to choose an existing line as the axis of rotation.

Number of Copies

This is the number of times the original shape is replicated. The more copies you make, the smoother the end result appears, but more copies also take longer to shade or edit. For most purposes you will probably want at least 10 copies per revolution (one copy every 36 degrees). The maximum number of copies is 198.

Span Angle

This is the number of degrees the object will sweep about its axis. For a complete, circular extrusion, enter 360°. If you are creating a spiral shape, you can enter more than 360° to achieve more than one revolution. For example, enter 1440 to get four complete revolutions.

Vertical Offset

This is the distance along the axis of rotation that the final copy is from the original. For normal, circular sweeps, this should be set to zero. If a value is used here, DesignCAD draws a spiral-shaped object instead of a circular object.

Vertical Offset Velocity

This is a ratio between the initial pitch of the offset and the final pitch of the offset. If a value of 5 is used, the final pitch will be 5 times greater than the initial pitch. If a value of 0 or 1 is used, the pitch will remain constant.

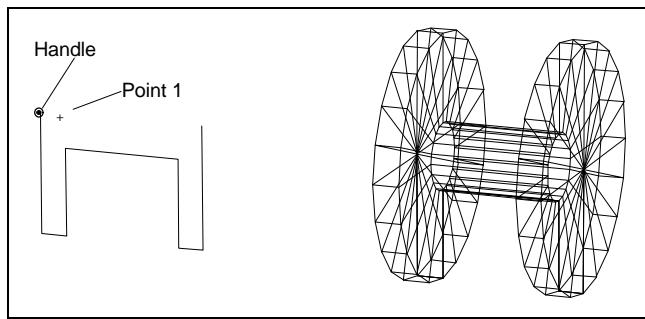
Radius Ratio

The distance from the selection handle to the axis is used for the sweep radius at the beginning of the sweep. The final radius for the sweep equals the beginning sweep radius multiplied by the Radius Ratio.

If a value of 1 is used for the Radius Ratio, the radius stays the same throughout the sweep. If a value greater than 1 is used, the distance from the axis to the extrusion increases throughout the sweep. If a value less than 1 is used, the distance from the axis to the extrusion decreases throughout the sweep.

Example: Draw a spool.

First, draw the outline or shape of the spool (shown below) and select it. Then choose the SWEEP command. Enter **20** in the COPIES box and **360** in the SPAN ANGLE field in the dialog box. Leave the offset as **0**. Choose the X-AXIS from the AXIS box. Next, set a point to the right of the handle. DesignCAD sweeps the connected lines and forms the spool.



Symbol Library Command

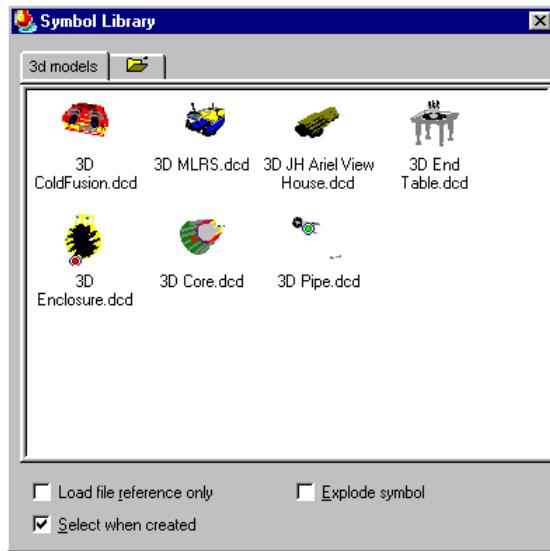
Menu:

FILE

Menu Command:

SYMBOL LIBRARY

The Symbol Library command works like the Load Symbol command, but it brings up a dialog box that shows small preview images or "thumbnails" of the symbols that are available in the different symbols libraries that come with DesignCAD.



You can move the dialog box by placing the cursor on the Symbol Library title bar, pressing and holding down the left mouse button, moving the mouse, and then releasing the left mouse button. You can also resize the dialog box by moving the cursor to an edge or corner of the dialog box (the cursor will turn into a two-way arrow when you have it placed correctly), pressing and holding the left mouse button, moving the mouse to resize the dialog box, and then releasing the left mouse button. Being able to move and/or resize the dialog box simplifies the task of making the areas in which you want to place the symbols visible.

Use the scroll bar on the right side of the dialog box to scroll through the various symbols. Click the tabs or use the scroll bar at the top of the dialog box (just under the Symbol Library title bar) to view related symbols in a different library.

DesignCAD 3D MAX allows you to view the DesignCAD 3D MAX files contained in a directory and all of the DesignCAD 3D MAX files in one layer of subdirectories of that directory. To view a different set of symbols click on the tab with a folder symbol on it. The Path dialog box opens. Select a new folder and click OK.

Load File Reference Only

This checkbox determines whether the drawing being added or "merged" with the current drawing will be saved as a part of the drawing or just linked by a reference.

If the option is not checked and the symbol is saved as part of the current drawing, the file size will be larger, and the drawing will not be updated when the symbol file is modified. The advantage is that the drawing will not be affected if the symbol file is deleted, or the drawing is opened on a computer that doesn't have a copy of the symbol file.

If the box is checked, the program adds an "insertion entity" to the drawing file. The symbol file is read every time the drawing is loaded. The symbol file must be present and in its original location. If the symbol file is modified, the change will be reflected in all the drawings using that symbol.

Select Object When Created

When this option is enabled, the symbol is automatically selected when it is added to the drawing.

Explode Symbol

When a referenced symbol is exploded, the symbol file is not read from disk when the drawing is loaded. A copy of the symbol is placed into the drawing instead of an insertion entity, just like a symbol that was loaded without using the Load File Reference Only option.

Placing a Symbol in Your Drawing

When you find the symbol that you want to use, there are two different methods for bringing it into the drawing:

1) move the cursor over the desired symbol, press and hold the left mouse button, drag the cursor to the desired location in your drawing, position the cursor and click the mouse button to drop the symbol into the drawing;

or,

2) move the cursor over the desired symbol, double-click the left mouse button, move the cursor to the desired location in your drawing, and click the left mouse button again to drop the symbol into the drawing.

Symbols are recognized as Groups (the entire symbol is one object, if you try to manipulate part of the symbol, the entire symbol is manipulated in the same way). To manipulate the different parts of the Symbol separately, select the symbol and then choose the GROUP EXPLODE command from the TOOLS menu. This will allow you to select and manipulate different parts of the symbol.

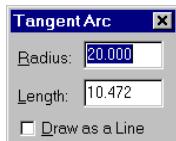
Tangent Arc Command

Menu:	DRAW
Submenu:	ARCS
Menu Command:	TANGENT ARC
Toolbox Icon:	
Point 1:	First point of line tangent to arc
Point 2:	Second point of line tangent to arc
Point 3:	Direction in which the arc is to be drawn

The Tangent Arc command lets you draw an arc with a specified radius by defining a line tangent for one end of the arc.

Using the Command

Select the TANGENT ARC command from the ARCS submenu of the DRAW menu. Enter the radius in the RADIUS box in the dialog box. Enter the length for the arc in the LENGTH box.

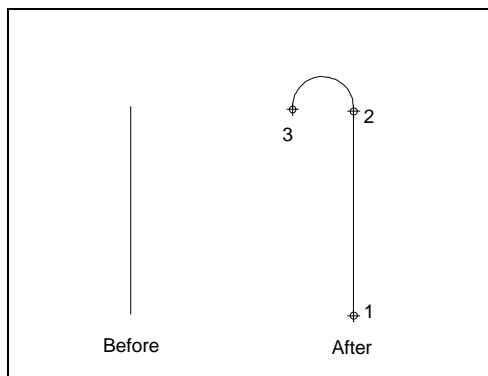


Note: The length of the arc is measured along the arc and is a linear measurement. It is not an angular or chord measurement.

Set the first point for the imaginary line to which the arc is to be drawn tangent. Set the second and final point for the line; this point will also serve as the starting point of the arc. (These first two points may be set on the endpoints of an existing line.) After the second point is set, a rubber-band arc is drawn and shows how the arc will be drawn. Move the mouse to determine the orientation of the arc. Set the endpoint for the arc.

Example: Draw an arc tangent to an existing line.

Select the TANGENT ARC command from the ARCS submenu of the DRAW menu. Enter **10** for the radius of the arc in the RADIUS box in the dialog box. Enter **40** for the length of the arc in the LENGTH box. Use the GRAVITY command to set a point on each end of the existing line. A rubber-band arc is drawn to show how the arc would be drawn if the current cursor position were used as the endpoint of the arc. Position the cursor and set a point for the arc's endpoint.



See Also: Arc Command, Arc (3-Point) Command, Arc (Center, Begin, End) Command, Arc (Endpoints, Center) Command, Arc (Radius, Begin-End) Command

Tangent Between Circles Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	TANGENT BETWEEN CIRCLES

Toolbox Icon:



Point 1: Point near the first circle.

Point 2: Point near the second circle

The Tangent Between Circles command draws a line tangent to two circles, two arcs, or an arc and a circle. There are four possible lines that can be drawn tangent to any two circles. The command draws a line closest to the two points that you set in the command.

Using the Command

Choose the TANGENT BETWEEN CIRCLES command in the Toolbox. Set a point on or near the first circle or arc. After the first point is set, a rubber-band line shows how the line will be drawn. Set a point on or near the second circle or arc. A line will be drawn tangent to the two circles or arcs.

Note: The Tangent commands do not work on circles that have been saved in vector form. These entities are not true circles. They are line entities approximating circles. Also, if you are not in 2-D Mode, the two lines must lie in the same plane.

Example: Draw a line tangent to two circles.

Draw two circles. Choose the TANGENT BETWEEN CIRCLES command. Set a point on or near the first circle or arc, close to the point where you want the tangent line to start. Set a second point near the second circle, close to the point where you want the tangent line to end. A line will be drawn tangent to the two circles.

Tangent From a Circle Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	TANGENT FROM A CIRCLE
Toolbox Icon:	
Point 1:	Point near circle
Point 2:	Endpoint for the tangent line

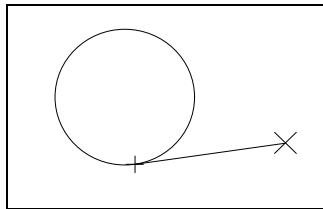
The Tangent From a Circle command draws a line tangent from a point on a circle, ellipse, arc, or elliptical arc to another point.

Using the Command

Choose the TANGENT FROM A CIRCLE command in the Toolbox. Set a point on or near the circle or arc. After the first point is set, a rubber-band line shows how the tangent will be drawn. Set a point for the end of the line on or near the line tangent to the circle or arc. The endpoint of the line will be even with the second point. A line will be drawn from the first point to the second point, tangent to the circle or arc.

Example: Draw a line tangent from a circle.

Choose the TANGENT FROM A CIRCLE command in the Toolbox. Set a point on or near the circle or arc. Notice as you move the cursor around the screen that a rubber-band line extends from the cursor to a point tangent to the circle. Set a point for the endpoint of the tangent line.



Tangent Snap Command

Menu:	POINT
Menu Command:	TANGENT SNAP
Toolbox Icon:	
Point 1:	Point near the circle or arc

The Tangent Snap command sets a point for a line or curve tangent to an existing circle or arc.

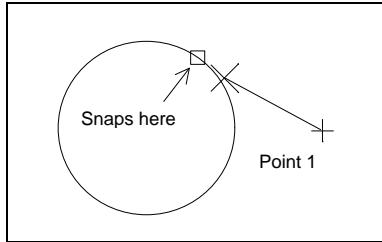
Using the Command

Set the first point for a line. Choose the TANGENT SNAP command. Move the cursor near an existing circle. A small box appears to show where the cursor will snap if the current cursor position is used. Set a point. The cursor snaps to the nearest tangent location and sets a point. Continue setting points for the line or press **Enter** to end the command and draw the line.

Note: If you are not in 2-D Mode, the line and circle must lie in the same plane.

Example: Draw a line tangent to a circle.

Choose the LINE command and set the first point. Choose the TANGENT SNAP command from the POINT menu. Move the cursor near the circle. A small box shows where the cursor will snap. Set a point to snap to the tangent of the line segment and the circle.



Tangent to a Circle Command

Menu:	DRAW
Submenu:	LINES
Menu Command:	TANGENT TO A CIRCLE
Toolbox Icon:	

Point 1: Endpoint for the tangent line
 Point 2: Point near the circle

The Tangent to a Circle command draws a line tangent from a point to a circle, ellipse, arc, or elliptical arc. There are two possible lines that can be drawn tangent to a circle from a single point. The line drawn is the one closest to the second point.

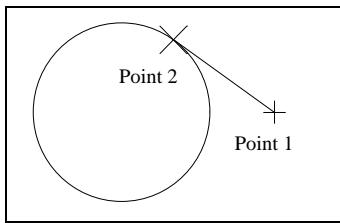
Using the Command

Choose the TANGENT TO A CIRCLE command in the Toolbox. Set a point for the beginning of the line. After the first point is set, a rubber-band line shows how the line will be drawn. Set a point on or near the circle or arc. A line will be drawn from the first point tangent to the circle or arc.

Note: If you are not in 2-D Mode, the line and circle must lie in the same plane.

Example: Draw a line tangent to a circle.

Choose the TANGENT TO A CIRCLE command in the Toolbox. Set a point for the beginning of the line. Move the cursor near the circle. A rubber-band line shows how the line will be drawn. Set the second point to draw the line.



Text Command

Menu: DRAW

Menu Command: TEXT

Shortcut Key: T

Toolbox Icon:

Point 1: Lower-left corner

Point 2: Lower-right corner (optional)

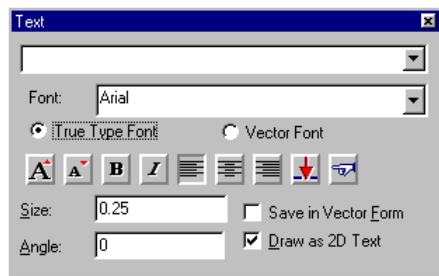
Point 3: Plane on which the text lies (optional)

Point 4: Slant and/or twisting of text (optional)

The Text command inserts a string of text into the drawing. This text may appear flat regardless of the viewing angle you are using or it can be placed at any 3-D orientation so its appearance varies depending on the current viewing parameters.

Using the Command

Choose the TEXT command in the Toolbox. Click the TEXT box in the dialog box and enter the text. The Text box keeps a history of the last 20 entries. To use this feature, click the down arrow on the right end of the box and highlight the entry to be used again.



Select the font for the text from the FONT list box. If the TRUE TYPE FONT option is chosen, only True Type fonts are available in the Font list box. If the VECTOR FONT option is chosen, only Vector fonts are available in the Font list box.

Click the INCREASE TEXT SIZE  button to increase the text size.

Click the DECREASE TEXT SIZE  button to increase the text size.

Click the BOLD  button to make the text bold.

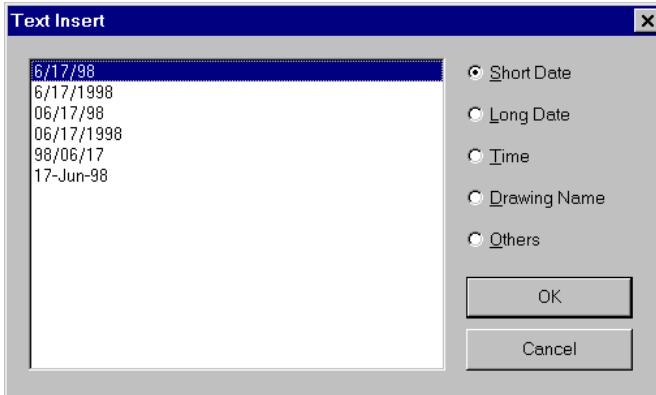
Click the ITALIC  button to italicize the text.

Click the LEFT JUSTIFY  button to align the text on the left.

Click the CENTER JUSTIFY  button to center the text.

Click the RIGHT JUSTIFY  button to align the text on the right.

Click the INSERT button  to display the Text Insert box.



Choose the kind of text you would like to have automatically inserted from the list of items to the right of the Text Insert dialog main window: Short Date for example. Using the mouse, click the format for the item from those displayed in the main window of the Text Insert dialog and then click the OK button.



Click the SAME AS button and then click on a line of text in the drawing to apply the same options to the new text.

You can enter the size for the text directly in the size box instead of using the Increase Text Size and Decrease Text Size buttons. If only one point will be set for the text, enter the angle for the text in the ANGLE box.

Note: If the Save in Vector Form option is checked, the text will be drawn as separate vector entities in the shape of each letter. Text drawn in this manner can not be edited as normal text.

Draw as 2D Text

If the Draw as 2D Text option is enabled, set a point for the bottom of the text. Next, Set a point to determine the angle of the text, or click the middle mouse button or press **Enter** to have the text drawn using the angle set in the Text Option box. The text will be drawn according to the points and the options selected. The text will appear flat regardless of the viewing angle being used.

If the Draw as 2D Text option is not enabled, you can set one to four points. If only one point is used, the size and angle of the text is determined by the text size and angle in the dialog box.

The second point is used to specify the length and angle of the text. The lower-right corner of the text will be positioned at the second point.

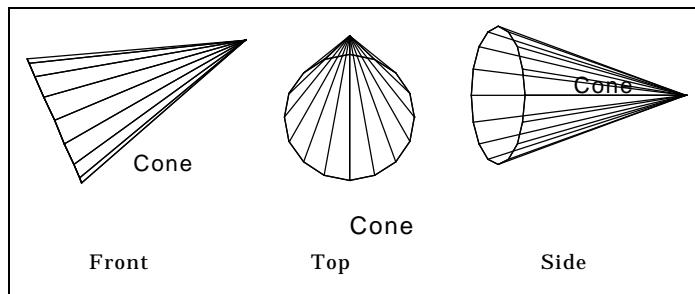
The third point, if used, defines the plane on which the text is to be drawn and partially determines the slant for the text. For example, if the third point is above the first, the text will be upright. If the third point is "behind" the first, then the text will be positioned to be read from above. It will be "flat" on the X-Z plane. If the third point is above and to the left of the first point, the top of the text will slant to the left.

The fourth point, if used, further defines the plane on which the text is to be drawn and partially determines the slant for the text. If four points are used, they essentially specify the positions of the corners of the text.

Example: Insert text into your drawing that will be visible in every view.

Select the **TEXT** command. Verify that the **DRAW AS 2D TEXT** option is checked. If it is not checked, click on it with the mouse. Enter the desired text in the **TEXT** box in the dialog box. A rubber-band text box follows the cursor on the drawing screen. Move the cursor to the place where you want the text inserted and set a point for the lower-left corner. The location of the second point determines the orientation of the text. Set these points where you like. Look at the drawing in

different views. Notice that while the text position may change, its orientation (facing you) never changes.



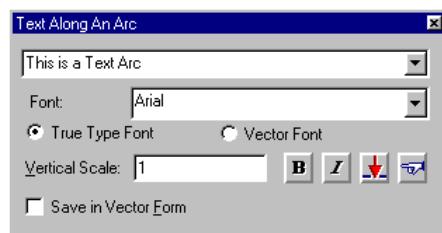
Text Arc Command

Menu: DRAW
 Menu Command: TEXT ARC
 Toolbox Icon: 
 Point 1: Start of the text arc
 Point 2: Center of the text arc
 Point 3: End of the text arc

The Text Arc command draws text along an arc. The arc is defined by a beginning point, a center point, and an endpoint.

Using the Command

Choose the TEXT ARC command in the Toolbox. Enter the text in the TEXT box.



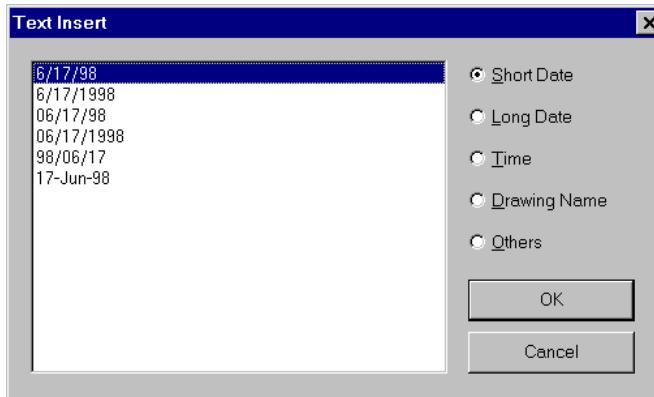
Select the font for the text from the FONT list box. If the TRUE TYPE FONT option is chosen, only True Type fonts are available in the Font list box. If the VECTOR FONT option is chosen, only Vector fonts are available in the Font list box.

Vertical Scale is the vertical scale factor of the text. A large value for Vertical Scale will make tall, thin letters. A small value will make short, wide letters. A Vertical Scale of 1.0 is normal.

Click the BOLD  button to make the text bold.

Click the **ITALIC**  button to italicize the text.

Click the **INSERT** button  to display the Text Insert box.



Choose the kind of text you would like to have automatically inserted from the list of items to the right of the Text Insert dialog main window: Short Date for example. Using the mouse, click the format for the item from those displayed in the main window of the Text Insert dialog and then click the **OK** button.

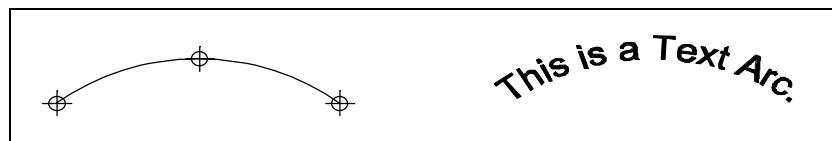
Click the **SAME AS** button  and then click on a line of text in the drawing to apply the same options to the new text.

Note: If the Save in Vector Form option is checked, the text will be drawn as separate vector entities in the shape of each letter. Text drawn in this manner can not be edited as normal text.

Set a point for the beginning of the text arc. Set a point for the center of the text arc. Set a point for the end of the text arc. The text will be drawn in an arc, beginning with the first point and ending with the last point.

Example: Draw a text arc.

Choose the **TEXT ARC** command in the Toolbox. Enter the text in the **TEXT** box. Set points for the beginning, center, and end of the arc. The text will be inserted along the arc.



See Also: **Text Block Command, Text Command**

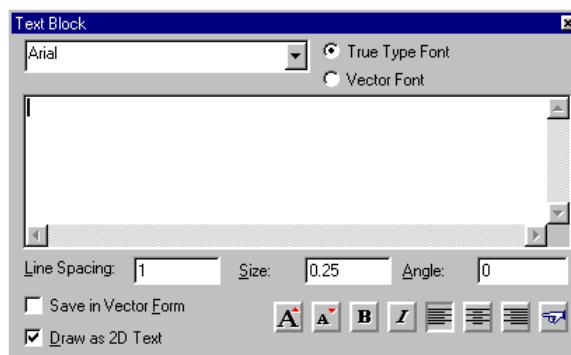
Text Block Command

Menu: DRAW
 Menu Command: TEXT BLOCK
 Toolbox Icon: 
 Point 1: Top of the Text Block
 Point 2: Point determining angle of the Text Block (optional)
 Point 3: Plane on which the text lies (optional)

The Text Block command allows drawing text to occupy multiple lines and sets the size, angle, and orientation of the text. Points determine the position and angle of the text in the drawing.

Using the Command

Choose the TEXT BLOCK command from the DRAW menu. Click the TEXT box in the TEXT BLOCK box and enter the text. When typing in the TEXT box press **Enter** to drop to the next line.



Select the font for the text from the FONT list box. If the TRUE TYPE FONT option is chosen, only True Type fonts are available in the Font list box. If the VECTOR FONT option is chosen, only Vector fonts are available in the Font list box.

Enter the desired spacing between lines in the LINE SPACING box.

You can enter the size for the text directly in the size box instead of using the Increase Text Size and Decrease Text Size buttons. If only one point will be set for the text, enter the angle for the text in the ANGLE box.

Note: If the Save in Vector Form option is checked, the text will be drawn as separate vector entities in the shape of each letter. Text drawn in this manner can not be edited as normal text.

Draw as 2D Text

If the Draw as 2D Text option is enabled, set a point for the upper-left corner of the text. Next, Set a point to determine the angle of the text, or click the middle mouse button or press **Enter** to have the text drawn using the angle set in the ANGLE option. The text will be

drawn according to the points and the options selected. The text will appear flat regardless of the viewing angle being used.

If the Draw as 2D Text option is not enabled, you can set one to three points. If only one point is used, the angle of the text is determined by the ANGLE option in the dialog box.

The second point is used to specify the angle of the text. The upper-right corner of the text will be positioned at the second point.

The third point, if used, defines the plane on which the text is to be drawn. For example, if the third point is above the first, the text will be upright. If the third point is "behind" the first, then the text will be positioned to be read from above. It will be "flat" on the X-Z plane.

Click the INCREASE TEXT SIZE  button to increase the text size.

Click the DECREASE TEXT SIZE  button to decrease the text size.

Click the BOLD  button to make the text bold.

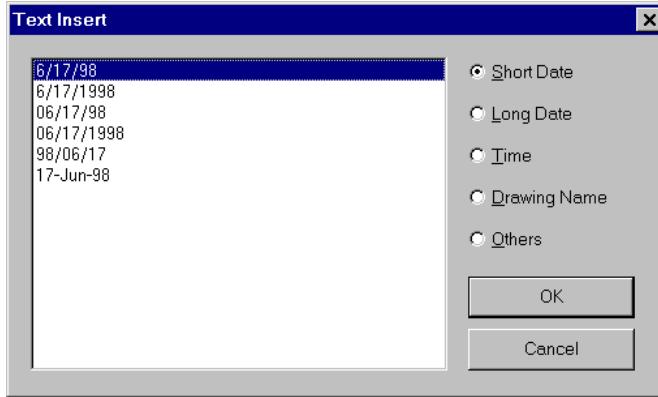
Click the ITALIC  button to italicize the text.

Click the LEFT JUSTIFY  button to align the text on the left.

Click the CENTER JUSTIFY  button to center the text.

Click the RIGHT JUSTIFY  button to align the text on the right.

Click the INSERT button  to display the Text Insert box.



Choose the kind of text you would like to have automatically inserted from the list of items to the right of the Text Insert dialog main window: Short Date for example. Using the mouse, click the format for the item from those displayed in the main window of the Text Insert dialog and then click the OK button.

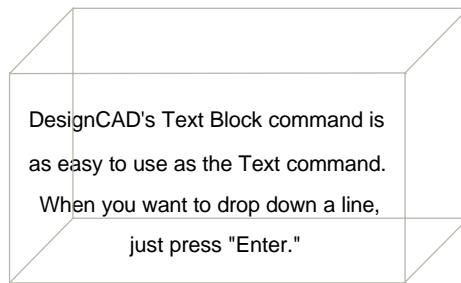


Click the SAME AS button and then click on a line of text in the drawing to apply the same options to the new text.

Set a point for the top of the text. Set a point to determine the angle of the text, or click the middle mouse button or press **Enter** to have the text drawn using the angle set in the Text Option box. The text will be drawn according to the points and the options selected.

Example: Insert a Text Block into a drawing.

Choose the TEXT BLOCK command. Click the TEXT box in the TEXT BLOCK box and enter the text you want to appear in your drawing. Press **Enter** to drop down a line. When you have finished the entry, set a point for the bottom of the text. Press **Enter** to insert the text.



Text Convert Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	TEXT CONVERT

The Text Convert command converts selected 2-D text to 3-D text and vice versa.

Using the Command

Draw some 2-D text using the TEXT command with the DRAW AS 2D TEXT option selected. Select the 2-D text. Now select the TEXT CONVERT command. The selected 2-D text is changed to 3-D text. Leave the text (now three-dimensional) selected. Select the TEXT CONVERT command again; the 3-D text is converted back to 2-D text.

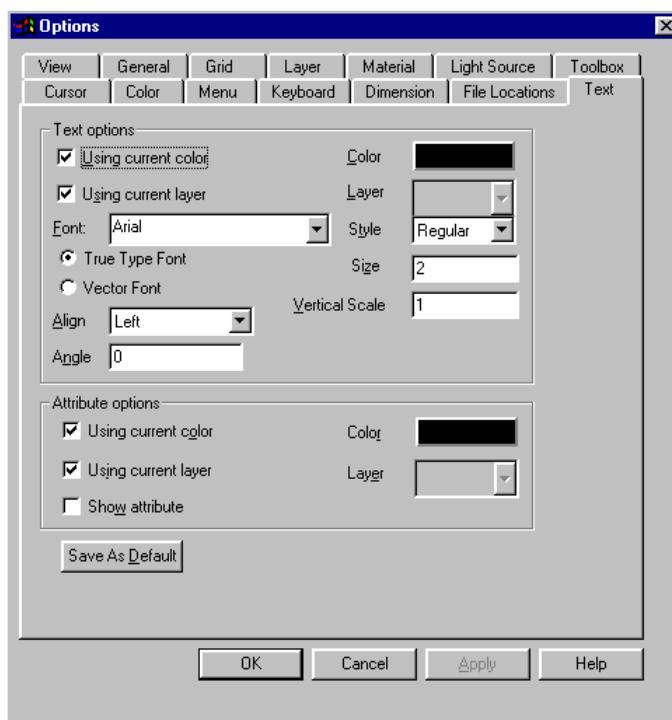
Text Options

Menu:	OPTIONS
Menu Command:	OPTIONS
Shortcut Key:	Q

In the Text Options folder, you can change the color and style of regular text and Attribute text for the different layers in your drawing.

Using the Command

Choose the OPTIONS command from the OPTIONS menu. Click the TEXT tab to bring up the Text Options folder.



Text Options

Using Current Color

When this option is selected, text is drawn with the current drawing color. If you are drawing with several different colors and want all text to be drawn with the same color, uncheck the USING CURRENT COLOR checkbox and click the COLOR box; the Color Palette appears. Click on the color you want to use for text and click OK.

Using Current Layer

This option determines the layer in which the text is drawn. When the Using Current Layer option is checked, all text (excluding Dimension and Attribute) is drawn in the active layer. To draw all of the text in the same layer of a multi-layer drawing, uncheck the USING CURRENT LAYER box and click the LAYER box. Select the layer you want normal text to be drawn in and click OK.

Font

This option determines which font is used. Either type the name of the font in the box or choose a font from the drop down list.

True Type Font

If this option is chosen, only True Type fonts are available in the Font list box.

Vector Font

If this option is chosen, only Vector fonts are available in the Font list box.

Align

This option determines the alignment of the text: to the left side of the page, the center of the page or the right side of the page.

Angle

This option determines the default angle at which the text will be placed if only one point is set.

Style

This option selects the style of the text: Regular, Bold, Italic or Bold Italic.

Size

This option determines the size of the text.

Vertical Scale

This option determines the height of the letters as compared to the width. Enter a large value for tall, thin letters or a small value for short, wide letters.

Attribute Options**Using Current Color**

When this option is selected, Attribute Text is drawn with the current drawing color. If you are drawing with several different colors and want all Attribute Text to be drawn with the same color, uncheck the USING CURRENT COLOR checkbox and click the COLOR box; the Color Palette appears. Click on the color you want to use for Attribute Text and click OK.

Using Current Layer

This option determines the layer in which the Attribute Text is drawn. When the Using Current Layer option is checked, all Attribute Text is drawn in the active layer. To draw all of the Attribute Text in the same layer of a multi-layer drawing, uncheck the USING CURRENT LAYER box and click the LAYER box. Select the layer you want Attribute Text to be drawn in and click OK.

Show Attributes

This option displays or hides the attributes in a drawing.

Save as Default

If you want to save the changes to the next session, select the SAVE AS DEFAULT option. Click OK when you are finished.

See Also: *Options Command*

Texture Mapping Command

Menu: TOOLS
 Menu Command: TEXTURE MAPPING

The Texture Mapping command copies a texture from an image file (bitmap, gif, jpeg, etc.) and applies that texture to one or more DesignCAD drawing objects. The next time the drawing is shaded using either Gouraud (Medium Quality) or Phong (Best Quality) method in the Shading Command, all items that have been assigned a texture will be shaded accordingly. Several jpgs are included with DesignCAD 3D MAX as sample textures and are located in the DesignCAD 3D MAX directory.

Using the Command

Select the object to be assigned a texture. Choose the TEXTURE MAPPING command from the TOOLS menu.



Load Texture

Click the LOAD TEXTURE button. The Select Texture dialog box appears. Make sure the directory that contains the image file (bitmap, jpeg, etc.) is listed in the Look In: box. Select the file from the area below the Look In: box or type the name in the FILE NAME: box. Click on the OK button.

Clear Texture

The Clear Texture button is used to remove all texture settings from the currently selected drawing objects.

Rotate 90°

The Rotate 90° button may be used to rotate the image file 90 degrees for the image's use as a texture. This button can be used two times to rotate the image 180 degrees and three times to rotate the image 270 degrees if necessary.

Mirror Horizontally

The Mirror Horizontally button "flips" the image so that all portions of the image that were on the left side will now be on the right and vice versa.

Mirror Vertically

The Mirror Vertically button "flips" the image so that all portions of the image that were on the top of the image will now be on the bottom and vice versa.

Customize Stretching

This option allows for the customization of the image's scale and aspect ratio. When the Customize Stretching option is selected:

- 1) the Tile option is disabled, the number of copies of the image that appear on a single surface is determined by the values set for Image Scale and Aspect Ratio;
- 2) the Seamlessly option is enabled and available for selection or de-selection; and
- 3) the Image Scale and Aspect Ratio options are enabled so their values may be set.

Image Scale

Enter the value for the scale at which the image is to be displayed. For example, a value of .5 will display the image at half of its original size and a value of 2 will display the image at two times its original size.

Aspect Ratio

Enter the value for the relationship of the vertical and horizontal scale factors at which the image is to be displayed. For example, a value of .5 will display the image so that the vertical scale is twice that of the horizontal scale, and a value of 2 will display the image so that the horizontal scale is twice that of the vertical scale.

Rotation Angle

This value displays the current rotation angle of the image being used as a texture. This value can be increased or decreased by clicking the ROTATE 90 button repeatedly.

Tile

This option lets several copies (or an array) of the image appear on a single surface. When this option is selected:

- 1) the Customize Stretching option is disabled, the number of copies of the image that appear on a single surface is determined by the values set for Horizontal Times and Vertical Times;

- 2) the Seamlessly option is enabled and available for selection or de-selection; and
- 3) the Horizontal Times and Vertical Times options are enabled so their values may be set.

Horizontal Times

Enter the number of copies of the image is to be displayed from left to right on the selected item(s).

Vertical Times

Enter the number of copies of the image to be displayed from top to bottom on the selected item(s).

Rotation Angle

This value displays the current rotation angle of the image being used as a texture. This value can be increased or decreased by clicking the ROTATE 90 button repeatedly.

Seamlessly

This option inverts one image everywhere two images meet, so the edges of the image copies will not be as noticeable.

Top		Top
Left	Left	Left
	Top	
Left	Top	Left
Top		Top
Top		Top
Left	Left	Left
	Top	

Mapping Mode

There are several Mapping Modes from which to choose. The default direction for the texture runs parallel to the Y axis; this direction can be changed for some of the Mapping Modes. Each Mapping Mode has a unique set of options available in the list box directly below the Mapping Mode list box. Each Mapping Mode and option combination works best with a different kind of entity. Some experimentation may be required to determine which Mapping Mode and option works the best for different items in a given drawing.

Spherical Mapping

As its name indicates, this particular mode works best on rounded three-dimensional surfaces. Unlike most of the other Mapping Modes, Spherical Mapping mode has a single

method: Center. Spherical Mapping mode takes the flat image and wraps and stretches it around the selected object. The direction for the texture determines the “poles” of the object. If the Y axis is said to run through the poles of the object, the top and bottom edges of the image are compressed to meet at these poles and are stretched to fit around the “equator” of the object.

Grid Patch Mapping

The Grid Patch mode divides the image into the number of surfaces the selected object has. Each division of the image is then assigned to a surface. This mode is recommended for items that have a flat face but have been extruded into a 3-D object or “grid.” This mode has two methods: Grid Only and Grid & Plane.

Grid Only

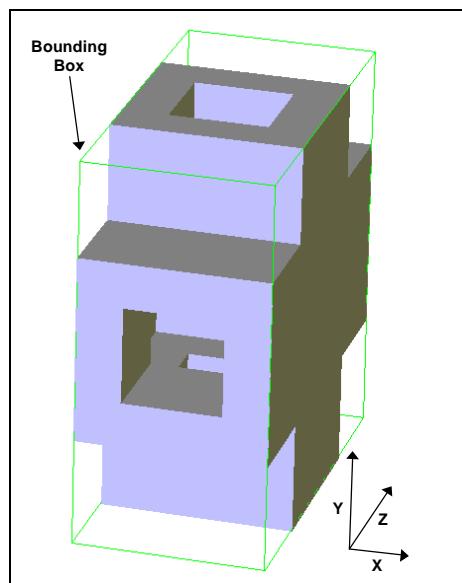
This method maps the selected texture to the “depth” or “grid” portion of the object (the portion of the object that is not the front “face”).

Grid & Plane

This method maps the selected texture to both the “face” and “grid” of the object.

Box Mapping

The Box Mapping mode uses a bounding box to map the texture to the selected objects. The bounding box is an invisible box that completely surrounds the selected objects. The front and back of this box run parallel to the XY plane. The left and right sides of this box run parallel to the YZ plane. The top and bottom of this box run parallel to the XZ plane.



This mode has six methods. Each of these six methods corresponds to a different side of the bounding box. The texture is placed parallel to the specified side of the bounding box and then projected onto the selected object(s).

Cylindrical Mapping

As its name indicates, this particular mode works best on cylindrically-shaped objects. Unlike most of the other Mapping Modes, Cylindrical Mapping mode has a single method: Center Axis. Cylindrical Mapping mode takes the flat image and wraps it around the selected object. The direction for the texture determines the “poles” of the object. If the Y axis is said to run through the “poles” of the object, the image is compressed to meet at these “poles.”

Plane Warp Mapping

This Mapping Mode should only be used on plane entities. Plane Warp Mapping mode has two methods. Default method maps the entity using as much of the image as possible without causing a substantial amount of distortion, the Set Boundaries and Change Boundaries options can be used so the entire image is used. The Cutoff method trims away portions of the image so that it has the same shape as the plane to which it is to be applied.

Default

When this method is chosen, two options appear below it: Set Boundaries and Change Boundaries. The Set Boundaries option can be checked to make the entire image be used when the drawing entity is mapped. The perimeter of the plane is calculated and the points on the plane that make the most sense *mathematically* are set as the four corners for the texture. The Change Boundaries button can be used to allow four points to be set in the drawing to reset the four corners of the texture.

Cutoff

The Cutoff method of Plane Warp Mapping trims away portions of the image so that it has the same shape as the plane to which it is to be applied. The direction of the texture may be set in the Rotate box.

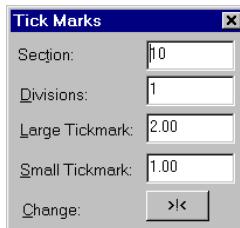
Tickmark Command

Menu:	DRAW
Menu Command:	TICKMARK
Toolbox Icon:	
Point 1:	Point on line, curve, or arc
Point 2:	Point indicating on which side of line, curve, or arc tick marks are to be drawn

The Tickmark command draws short lines, or tick marks, along a line, curve, ellipse, or circle entity. These tick marks are spaced at specified intervals along the entity.

Using the Command

Choose the TICKMARK command in the Toolbox. Enter the number of sections of large tick marks in the SECTIONS box in the dialog box. This determines the number of large tick marks to be evenly measured out along the length of the item being marked.

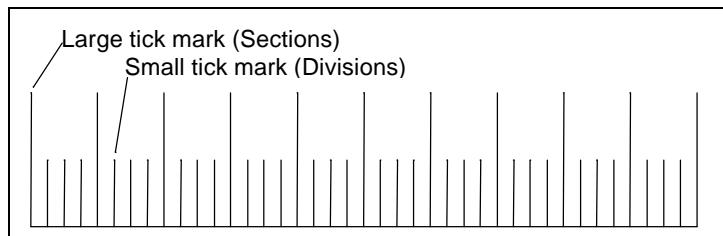


 If you would rather determine the exact distance between large tickmarks click the Even/Exact Switch button in the dialog box. The Sections box will be replaced with the Mark Distance box. This box lets you specify the exact distance between the large tick marks. To switch back to the Sections box, click the button again.

Enter the number of divisions of tick marks in the DIVISIONS box. This divides each section up with small tick marks. Enter the large tick mark size in the LARGE TICKMARK box. Enter the small tick mark size in the SMALL TICKMARK box. Set a point on the existing line. Set a point to indicate on which side of the line the tick marks will be drawn. If both points are set in the same location, the tick marks will be centered on the line. Tick marks will be drawn at even intervals along the line, on the side of the second point.

Example: Draw tick marks along a 10 unit line.

Draw a 10-unit-long line using the Line command. Choose the TICKMARK command in the Toolbox. Enter 10 in the SECTIONS box in the dialog box. Enter 4 in the DIVISIONS box. Set a point on the line. Decide on which side of the line the tick marks should be placed. Set another point to that side of the line. The tick marks will be drawn as follows: 10 one-unit sections, 4 divisions to each section.



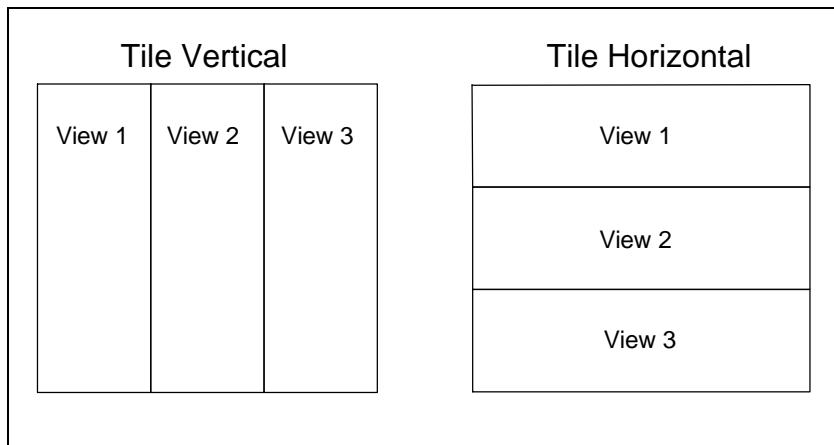
Tile Horizontal Command

Menu:	WINDOW
Menu Command:	TILE HORIZONTAL

The Tile Horizontal command organizes your open windows by arranging them horizontally across the screen. Each window takes up the same amount of space on the screen.

Using the Command

Select the TILE HORIZONTAL command from the WINDOW menu. The open drawing windows are stacked on top of each other.



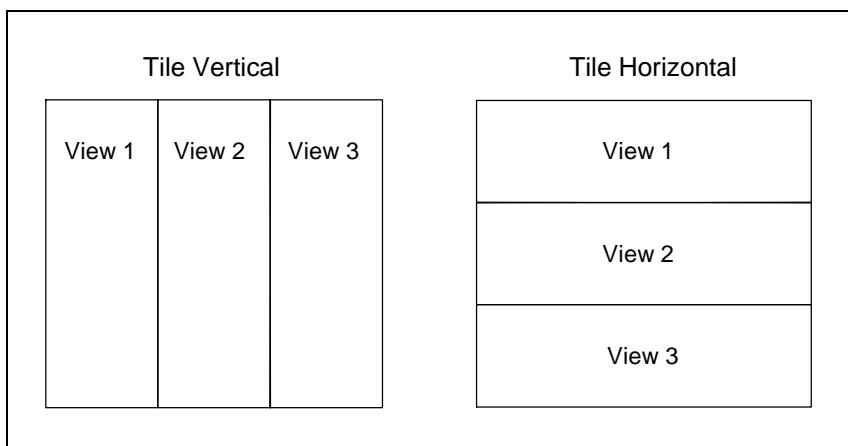
Tile Vertical Command

Menu: WINDOW
Menu Command: TILE VERTICAL

The Tile Vertical command organizes your open windows by arranging them vertically across the screen. Each window takes up the same amount of space on the screen.

Using the Command

Select the command from the WINDOWS menu. The open drawing windows will be placed side-by-side.



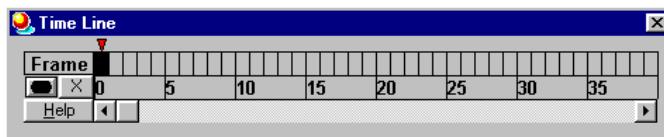
Time Line Command

Menu: ANIMATION
 Menu Command: TIME LINE

The Time Line command is a toggle that hides or shows the Animation Time Line depending on the dialog's current status.

Using the Command

While in Animation Mode, select the TIME LINE command from the ANIMATION menu. If the Animation Time Line was visible, it will be hidden. If the Time Line was hidden, it will be made visible.



See Also: Animation Mode Command, Control Panel Command

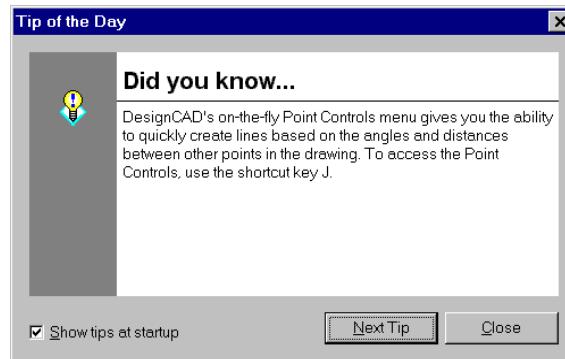
Tip of the Day Command

Menu: HELP
 Menu Command: TIP OF THE DAY

The Tip of the Day command opens the Tip of the Day dialog box. In this dialog box, tips can be viewed and the Show Tips at Startup option can be enabled or disabled.

Using the Command

Choose the TIP OF THE DAY command from the HELP menu. The Tip of the Day dialog box appears with a tip that will help you work with DesignCAD more efficiently. To view another tip click on the NEXT TIP button. Click the CLOSE button to close the Tip of the Day and return to DesignCAD.



If the Show Tips at Startup option is checked, the Tip of the Day dialog box will appear when you start DesignCAD. To disable this option, uncheck the Show Tips at Startup option by clicking on the checkbox.

Toolbar Menu Command

Menu:	VIEW
Menu Command:	TOOLBAR MENU

The Toolbar Menu command is used to show or hide toolboxes on the drawing screen, depending on the status of the toolbox at the time the command is chosen.

Using the Command

Choose the TOOLBAR MENU command from the VIEW menu. A list of available toolboxes appears. Toolboxes that are currently visible have a check mark beside their names. Toolboxes that are currently hidden do not have a check mark beside their names. Click on the name of the toolbar you wish to show or hide.

Toolbar Command

Menu:	TOOLS
Submenu:	CUSTOMIZE
Menu Command:	TOOLBAR

The Toolbar command is a shortcut method of bringing up the Toolbox folder of the Options file box.

Using the Command

Choose the TOOLBAR command from the CUSTOMIZE submenu of the TOOLS menu. The Options file box is displayed with the Toolbox folder showing. For a complete listing of the options available in this folder, see Toolbox Options.

See Also: *Toolbox Options*

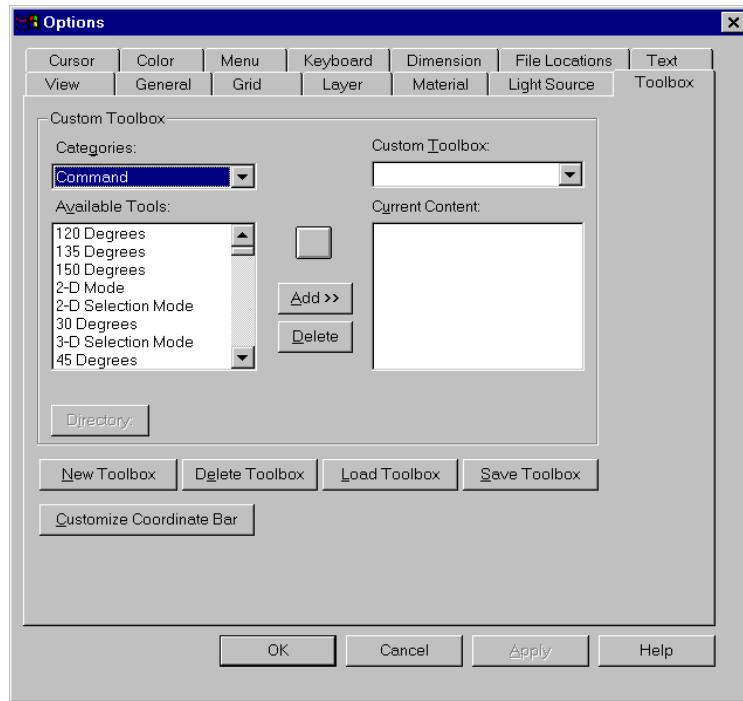
Toolbox Options

Menu:	OPTIONS
Menu Command:	OPTIONS

The Toolbox Options folder can be used to configure toolbox settings.

Using the Command

Choose the OPTIONS command. Then click on the Toolbox tab to bring up the TOOLBOX OPTIONS folder.



Custom Toolbox

Categories

You can choose to insert commands, macros, or BasicCAD programs into the Custom Toolbox. Each toolbox can hold as many as 48 items.

Available Tools

Choose which tools you want to add to the Custom Toolbox from the list.

Current Content

This is a list of tools which are currently in the Custom Toolbox.

Icon

This button displays the icon for the highlighted tool.

Add

This button inserts a selected tool into the Custom Toolbox.

Note: You can also insert a command icon into a Custom Toolbox by holding down **Ctrl** while clicking and dragging the icon from another tool box into the Custom Toolbox.

Delete

This button removes a selected tool from the Custom Toolbox.

Directory

This button allows you to choose the directory for macros and BasicCAD files.

New Toolbox

This button prompts you for a Custom Toolbox name and creates a new toolbox to which you may add commands.

Delete Toolbox

This button deletes the currently selected Custom Toolbox.

Load Toolbox

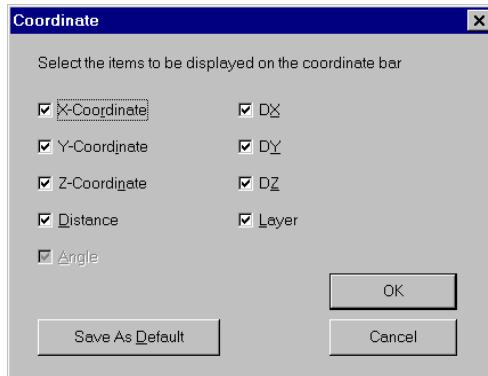
This button loads the .dct file for a Custom Toolbox.

Save Toolbox

Click the SAVE TOOLBOX button to save the toolbox settings as an external file. This file may then be used for one or more DesignCAD Workspaces.

Customize Coordinate Bar

This button opens the Coordinate dialog box.



Note: The Angle field is only available for 2-D Mode.

This box is used to determine what values are displayed in the Coordinate bar. Click on the box to the left of the item you want to display. A check mark will be placed in the box to indicate that it is visible. Click again to deselect and turn off the item. The check mark is removed to indicate that the item is hidden. Click SAVE AS DEFAULT to save the changes to the next drawing session. Click the OK button to accept the changes and return to the Toolbox Options folder. Click the CANCEL button to return to the Toolbox Options folder without changing anything.

See Also: *Load DesignCAD Workspace Command, Save DesignCAD Workspace Command*

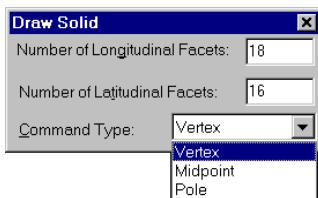
Torus Command

Menu:	SOLIDS
Menu Command:	TORUS
Toolbox Icon:	
Point 1:	Center of the torus
Point 2:	Center of the cross-section
Point 3:	Radius of the cross-section

The Torus command draws a solid torus.

Using the Command

Choose the TORUS command. You can specify the number of sides or facets around both the longitude and latitude of the torus in the FACETS ALONG LONGITUDE AND LATITUDE boxes in the dialog box. The more facets the torus has, the smoother it appears.



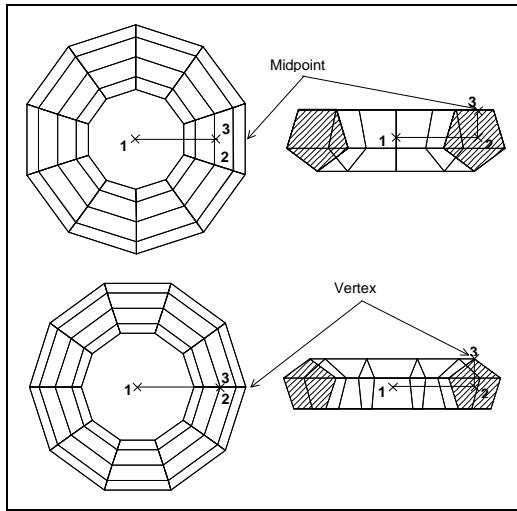
The command requires three points to be set: one for the center of the torus; a second for the center of the cross-section; and a third for the radius of the cross-section. You can choose whether the second and third points will be set at a vertex of the torus' perimeter and cross-section or a midpoint of its perimeter and cross-section.

If you choose VERTEX, the center for the cross-section of the torus is inscribed by a circle of the radius defined by the distance from Point 2 to Point 1. If you choose MIDPOINT, the center for the cross-section circumscribes a circle of that radius. Also, if you choose VERTEX, the cross-section, of the torus is inscribed by a circle of the radius defined by the distance from Point 3 to Point 2. If you choose MIDPOINT, the cross-section of the torus circumscribes a circle of that radius. This is normally not significant, but it can be important for some precision drawings.

Example: Draw a torus in your drawing.

Select the TORUS command. Enter 5 in the LONGITUDE box (left) in the dialog box. Enter 10 in the LATITUDE box (right). Choose VERTEX or MIDPOINT. Set a point for the center of the torus. Move the cursor out along the Y axis and set the second point for the center of the cross-section. Next, set a third point for the radius of the cross-section.

Hint: Points 1, 2 and 3 should not lie on a straight line.



Trim Between Two Lines Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	TRIM BETWEEN TWO LINES
Toolbox Icon:	
Point 1:	Line to be trimmed
Point 2:	First intersecting line
Point 3:	Second intersecting line

The Trim Between Two Lines command erases a segment of a line between its intersection with two other lines.

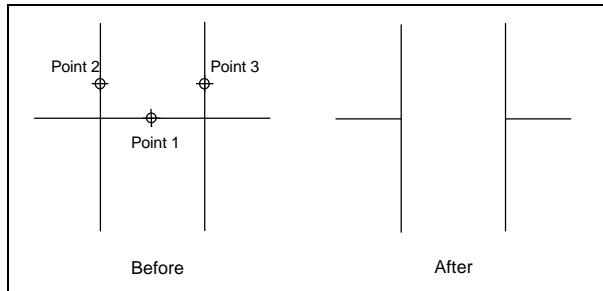
Using the Command

Choose the TRIM BETWEEN TWO LINES command from the Toolbox. Set a point on the line to be trimmed. Set a point on the first intersecting line. Set a point on the second intersecting line. The line is trimmed between the other two lines.

Note: In 3-D mode, all the lines must lie in the same plane for this command to trim them. In 2-D Mode, the command will trim the lines along their XY projections.

Example: Trim a line between two other lines.

Draw three lines in the shape of a letter "H," with the horizontal line passing through both vertical lines. Select the TRIM BETWEEN TWO LINES command from the Toolbox. Set a point on the horizontal section. Then set points on the vertical lines. The part of the horizontal line between the vertical lines is removed.



Trim Double Lines Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: TRIM DOUBLE LINES
 Toolbox Icon: 

Point 1: Corner of area containing lines to be trimmed
 Point 2: Opposite corner of area containing lines to be trimmed

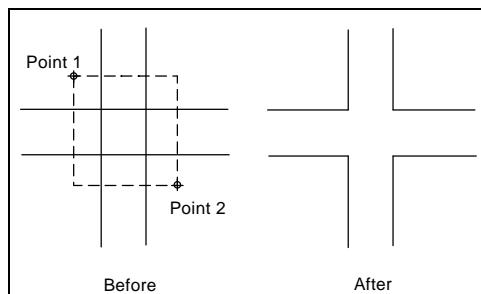
The Trim Double Lines command "trims" two sets of parallel lines at their intersection. This command only works in 2-D Mode.

Using the Command

Choose the TRIM DOUBLE LINES command from the Toolbox. Next, set a point in the corner of the area containing the lines to be trimmed. Set a point in the opposite corner of the area containing the lines to be trimmed. The two sets of lines will be trimmed to their intersection. Depending on the configuration of the lines, they will be trimmed to a "+," a "T," or an "L" shape.

Example: Trim double lines in your drawing.

Draw four lines (two vertical, two horizontal) so that they look like a number symbol (#). Select the TRIM DOUBLE LINES command from the TRIM/EXTEND submenu of the EDIT menu. Set a point for a corner of the area to be trimmed. Set another point for the opposite corner of the area to be trimmed. The two sets of lines will be trimmed at their intersection.



Trim Multiple Lines Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: TRIM MULTIPLE LINES
 Point 1-n: Lines to be trimmed
 Point n+1: Line to trim against

The Trim Multiple Lines command trims multiple lines at their intersection with another line. This command trims arc, curve, and line entities.

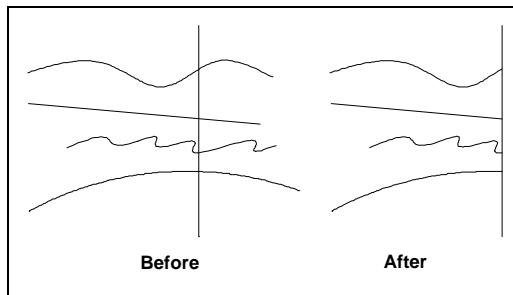
Using the Command

Select the TRIM MULTIPLE LINES command. Set a point on each line to be trimmed. Then set a point on the line to trim against. Press **Enter**. The lines will be cut off at their intersection with the line on which the last point was set.

Note: If you are in 2-D Mode, you can trim two lines that never meet by trimming their projections on the XY plane. Of course, when you return to 3-D mode, they still don't meet.

Example: Trim several lines, curves, and arcs against a line they all intersect.

Select the TRIM MULTIPLE LINES command. Set a point on all of the entities to be trimmed. Then set a point on the trim line. Press **Enter**. DesignCAD trims the lines back to their intersection with the trim line.



See Also: Trim One Line Command, Trim Two Lines Command

Trim One Line Command

Menu: EDIT
 Submenu: TRIM/EXTEND
 Menu Command: TRIM ONE LINE
 Shortcut Key: **E**
 Toolbox Icon: 
 Point 1: Line to be trimmed
 Point 2: Line to trim against

The Trim One Line command can be used to trim a line to its intersection with another line. This command works with lines, circles, and arcs.

Using the Command

Choose the TRIM ONE LINE command. Set a point on the line to be trimmed and a point on the line to be trimmed against. The first line is cut off at its intersection with the second. If the first line does not intersect the second, it is extended until they meet.



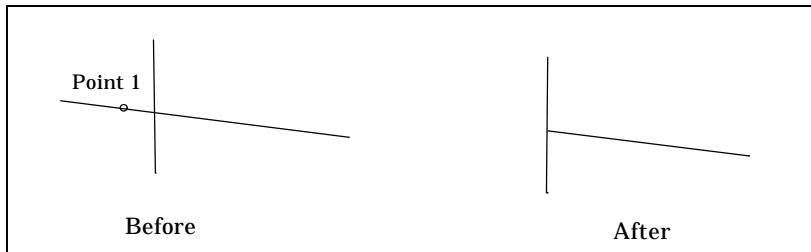
Trim Shorter End

If you check this box, the shortest end of the line is always trimmed off. If this box is not checked, the part of the line that you set the first point on is kept, and the opposite end is removed.

Note: If you are in 2-D Mode, you can trim two lines that never meet by trimming their projections on the XY plane.

Example: Trim the short end of a line that intersects with another.

Select the TRIM ONE LINE command and click on the TRIM SHORTER END checkbox. Set a point on the line that you want trimmed. Then set a point on the line to be trimmed against. DesignCAD trims the line back to its intersection with the other line.



See Also: Trim Two Lines Command

Trim Two Lines Command

Menu:	EDIT
Submenu:	TRIM/EXTEND
Menu Command:	TRIM TWO LINES
Shortcut Key:	Ctrl+E
Toolbox Icon:	
Point 1:	First line to be trimmed
Point 2:	Second line to be trimmed

The Trim Two Lines command trims two lines at their intersection, forming a clean corner with no overlap. This command trims only lines and arc entities.

To use the Trim Two Lines command, set a point on each line to be trimmed. The lines will be cut off at their intersection. If the lines do not intersect, they will be extended to the point of intersection.



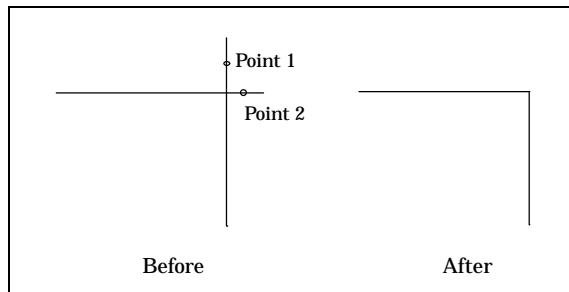
Trim Shorter End

If this box is checked, the shortest ends of the two lines will always be trimmed. If it is not checked, then place the points on the portions of the lines that you wish to keep, and the opposite ends will be trimmed away.

Note: If you are in 2-D Mode, you can trim two lines that never meet by trimming their projections on the XY plane. Of course, when you return to 3-D mode, they still don't meet.

Example: Trim the short ends of two intersecting lines.

Select the TRIM TWO LINES command and click on the TRIM SHORTER END checkbox. Then set points on both lines. DesignCAD trims the lines back to their intersection.



See Also: Trim One Line Command

Truncated Cone Command

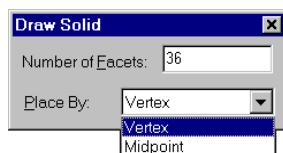
Menu:	SOLID
Menu Command:	TRUNCATED CONE
Toolbox Icon:	
Point 1:	Center of the base of the cone
Point 2:	Edge of the base of the cone
Point 3:	Height of the cone
Point 4:	Edge of the top of the cone

The Truncated Cone command draws a solid truncated cone.

Using the Command

Set a point for the center of the base of the cone, a second point at the edge of the base, and a third point for the cone height. Next, move the cursor inward toward the center and set the fourth point for the edge of the truncated point of the cone.

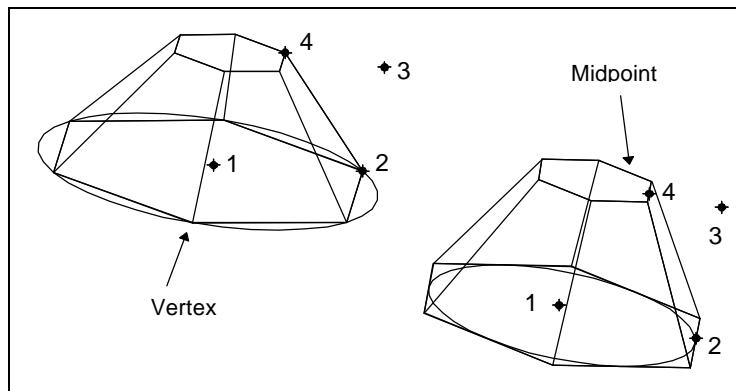
You can specify the number of sides or facets around the cone in the NUMBER OF FACETS field in the dialog box.



You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Points 2 and 4. If you choose VERTEX, the radius of the cone is inscribed by a circle of that radius. If you choose MIDPOINT, the radius of the cone circumscribes a circle of that radius. This is normally not significant, but it can be important for some precision drawings.

Example: Draw a truncated cone.

Select the TRUNCATED CONE command. Next set a point for the center of the base. Move the cursor out along the Y axis and set the second point for the radius of the cone. Now move the cursor up until the cone is the desired height and set the third point. Move the cursor back in along the Y axis and set the fourth point for the radius of the cone's truncated tip. The cone is inserted into the drawing.



See Also: Cone Command

Tube Command

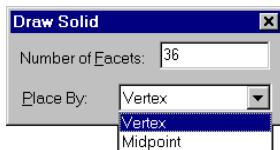
Menu:	SOLIDS
Menu Command:	TUBE
Toolbox Icon:	

- Point 1: Center of the tube
- Point 2: Radius 1 (inner or outer)
- Point 3: Length of the tube
- Point 4: Radius 2 (outer or inner)

The Tube command draws a solid tube.

Using the Command

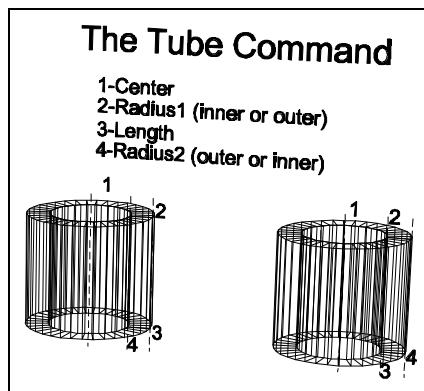
You can specify the number of sides or facets around the tube in the NUMBER OF FACETS box in the dialog box.



You can also choose whether the midpoint or vertex of the facets will be located at the radius defined by Point 2. If you choose VERTEX, the inner/outer radius of the tube is inscribed by a circle of that radius. If you choose MIDPOINT, the inner/outer radius of the tube circumscribes a circle of that radius.

Example: Draw a tube in your drawing.

Select the TUBE command. Set a point for the center of the tube. Move the cursor along the Y axis and set the second point for the first radius of the tube; in this example, it will be the inner radius. Next, move the cursor up (or down) until the tube is the desired length, and set the third point. Move the cursor out along the Y axis again and set the fourth point for the second radius; in this example, it is the outer radius. The tube will be inserted into the drawing.



Undo Command

Menu:	EDIT
Menu Command:	UNDO
Shortcut Key:	Ctrl + Z
Toolbar Icon:	

The Undo command cancels the most recent drawing action. It can be used repetitively to "back out" of a series of commands, as it is always negating the previous drawing action. You can "back up" to the point where the drawing was last saved.

See Also: [Redo Command](#)

Units Command

Menu: DIMENSION

Menu Command: UNITS

Shortcut Key: U

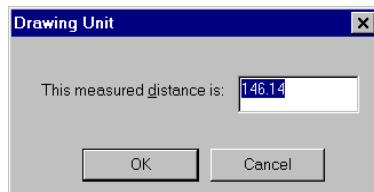
Point 1: First point on distance to measure.

Point 2: Second point on distance to measure.

The Units command can be used to measure the distance between two points and, if you choose, to change the distance. This command can also be used to change the units of measurement in the drawing or to set up the initial drawing space.

Using the Command

Choose the UNITS command. Set two points on the screen for a known distance. The Units box is displayed. You can accept the distance or change it in the THIS MEASURED DISTANCE IS box. Then click OK.



When you change the Drawing Units, it changes the entire coordinate system of the drawing, including any dynamic dimensions in the drawing.

Example: Set your drawing screen so it is 100 Units wide.

After you have opened a new drawing, select the UNITS command. Set a point at the left edge of the screen and another at the right edge. Then enter **100** as the new measurement in the field in the UNITS dialog box.

Use Resizing Handles Command

Menu: OPTIONS

Menu Command: USE RESIZING HANDLES



Toolbar Icon:

When the Use Resizing Handles command is enabled, items that are selected have scaling nodes or resizing handles around them. These nodes may be used to scale the selection along a single axis, both axes in 2-D Mode, or all three axes in 3-D Mode.

Using the Command

Select the items to be scaled or zoomed. Select the USE RESIZING HANDLES command from the OPTIONS menu. Resizing handles appear on and around the selection. These nodes may be used by placing the cursor over one of them, clicking the left mouse button, dragging the cursor to make the selection larger or smaller, and then clicking the left mouse button a second time to release the selection. The resizing handles will remain visible until the Use Resizing Handles command is selected again to turn the resizing handles off.

If you just use the left mouse button, the edge opposite the resizing handle you chose maintains its original location. To scale the selection by its center, press and hold down the **Ctrl** key when you click on the resizing handle, and release the **Ctrl** key after you click the mouse.

To zoom the selection (scale the selection equally with respect to all axes at once, see the Selection Zoom Command entry in the "Command Reference" section of this manual), press and hold down the **Shift** key when you click on the resizing handle, and release the **Shift** key after you click the mouse.

To zoom the selection by its center, press and hold down the **Ctrl+Shift** keys when you click on the resizing handle, and release the **Ctrl+Shift** keys after you click the mouse.

Vector Convert Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	VECTOR CONVERT
Point 1-n:	Entities to be changed to line entities

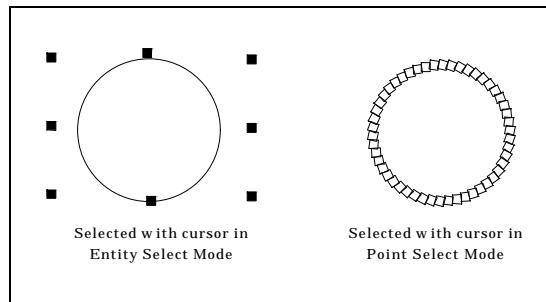
The Vector Convert command converts selected entities such as arcs, circles, curves, and planes to vector entities.

Using the Command

Select the entity to be converted. Choose VECTOR CONVERT. The entity is converted to a series of vectors, or short line segments. The new entity has more points than the original but looks the same.

Example: Convert a circle to vectors.

First, select the entity you want to convert to vectors. Now choose the VECTOR CONVERT command. DesignCAD automatically converts the entity to vectors. Although the entity does not change appearance, you can see the effect of conversion to vectors by choosing Point Select Mode and selecting the entity.



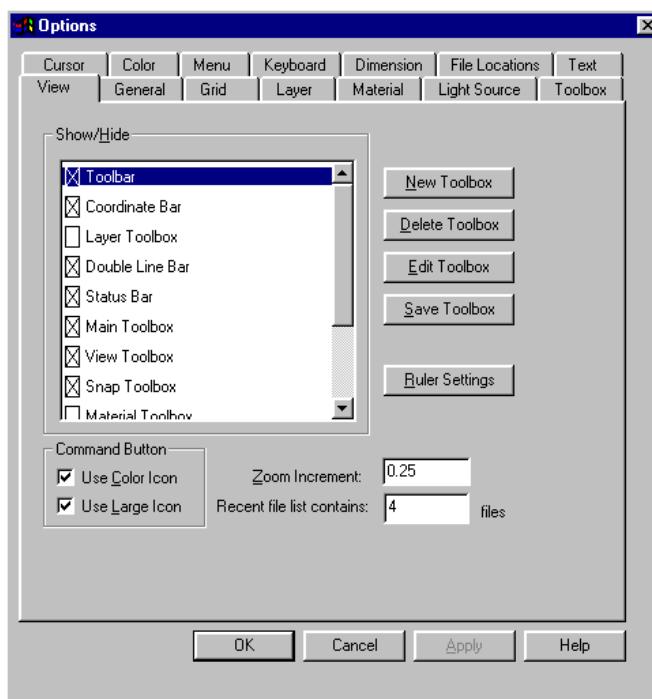
View Options

Menu: OPTIONS
 Menu Command: OPTIONS

The View Options folder allows you to choose which bars and toolboxes are displayed during the drawing session.

Using the Command

Choose the OPTIONS command, and then click on the VIEW tab to bring up the View Options folder.



Show/Hide

To activate a bar or toolbox, click the checkbox beside its name.

New Toolbox

This option creates a new Custom Toolbox. You may create as many as eight Custom Toolboxes. Each one may contain as many as 48 command, macro, or BasicCAD icons.

Delete Toolbox

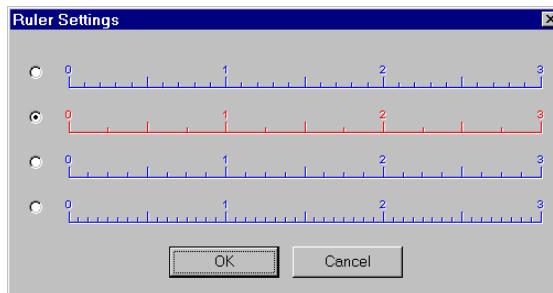
Deletes a Custom Toolbox.

Edit Toolbox

This option brings up the TOOLBOX options folder, with the current toolbox selected. Only Custom Toolboxes can be edited.

Ruler Settings

This option brings up the Ruler Settings dialog box which allows you to set the ruler divisions.



Hint: The Ruler Command, which displays or disables the ruler, is available under the View Command on the Main Menu. The ruler is available only in 2-D mode.

Use Color Icon

When this option is checked, icons are displayed in color. If this option is not checked, the option is disabled and the icons are displayed in grayscale.

Use Large Icon

When this option is checked, large icons are displayed. If this option is not checked, the option is disabled and small icons are displayed.

Zoom

This text field determines what zoom factor will be used for the Zoom commands. The default value is .25 (or 25 percent).

Recent File List Contains:

This option determines how many recent files will be listed in the File menu.

Volume Command

Menu: DIMENSION

Submenu: INFO

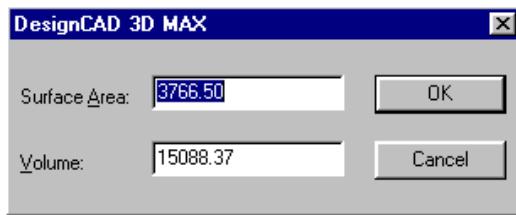
Menu Command: VOLUME

Point 1: Object for which to calculate volume

The Volume command calculates the volume and surface area of a Solid object.

Using the Command

Choose the Volume command, and then set a point on the Solid. DesignCAD does the rest!



Example: Determine the volume of an object in a drawing.

Select the VOLUME command. Set a point on the object. DesignCAD displays the volume of the object in a dialog box, along with the surface area.

VRML Command

Menu:	ANIMATION
Submenu:	EXPORT
Menu Command:	VRML

The VRML command exports an animation or walk through produced in DesignCAD as a VRML file so it can be used in a Web Page.

Using the Command

Record or open an existing animation or walk through template. Create a VRML file from your template by clicking ANIMATION | EXPORT | VRML. Give the file a name, then click SAVE.

See Also: *Animation Mode Command, AVI Command, Walk Through Mode Command*

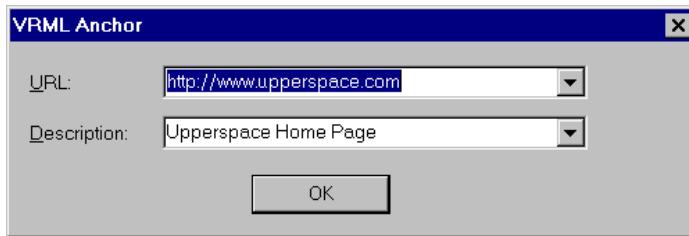
VRML WWW Anchor Command

Menu:	TOOLS
Menu Command:	VRML WWW ANCHOR

The VRML WWW Anchor command lets you add a link for the World Wide Web to any non-animated VRML file.

Using the Command

Open a file that you are going to export as a non-animated VRML file. Select the portion you want to use as a link. Select the VRML WWW ANCHOR command.



Enter the destination URL in the URL: box. In the DESCRIPTION: box, enter the description you want displayed when the user places the cursor over the graphic. Export the VRML with the EXPORT command.

Walk Through Mode Command

Menu:	ANIMATION
Menu Command:	WALK THROUGH MODE

The Walk Through Mode command works as a toggle to turn Walk Through Mode on and off. Walk Through Mode can be used step a viewer around a DesignCAD drawing.

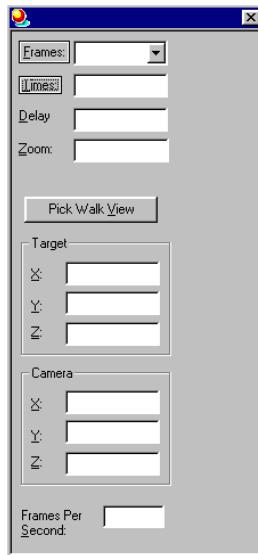
Like Animation Mode, Walk Through Mode can be used to produce AVI video files that can be played by Windows' Media Player and other video player applications. Walk Through Mode will also produce VRML files, which can be used to walk viewers through a drawing on the Internet.

The main difference between Walk Through Mode and Animation Mode is that Walk Through Mode is only used to change the view of a drawing; items in the drawing cannot be moved or rotated independent of one another. A series of view angles are changed slightly from one frame to another. When all the frames are displayed in rapid succession, the view angle of the viewer moves smoothly around the drawing.

Using the Command

With a drawing open, choose WALK THROUGH MODE from the ANIMATION menu. The Control Panel and Walk Through dialog boxes appear.





To set the first keyframe, click on the **FRAMES:** button and select ADD FRAME from the menu that appears. Set the amount of time (in seconds) to pass between this first frame and the next one in the **TIME:** box. Set the amount of time to pass (in seconds) before movement toward the next keyframe begins in the **DELAY:** box. You may also want to change the apparent size of the drawing by entering a new value in the **ZOOM:** box. Then use the **TARGET** and **CAMERA** areas of the dialog box. The **TARGET AREA** is used to specify what area of the drawing is being focused on (displayed in the center of the view window). The **CAMERA** area is used to specify the viewing location and thus controls the view angle and distance. Finally set the number of frames per second to be inserted between this frame and the next one in the **FRAMES PER SECOND:** box.

To add the next keyframe, click on the **FRAMES:** button and select ADD FRAME from the menu that appears. Change the amount of time to pass between this frame and the next one or leave it alone so it will be the same as the amount of time between the first frame and this one. Change any or all of the other options. When the view is the way you want it for the second keyframe, add another one.

Repeat this series of actions as many times as necessary to complete your walk through. If you make a mistake select the keyframe you want to change in the **FRAMES:** list box and change the values.

Playing and Saving A Walk Through Sequence

-  To rewind your walk through, click on the **REWIND** button.
-  Click on the **BACK ONE FRAME** button to move the walk through backward one frame.
-  Click the **STOP** button to stop a walk through that is playing.
-  Click the **PLAY** button to play and examine the walk through.

- ▶ Click the FORWARD ONE FRAME button to move the walk through forward one frame.
- ▶▶ To fast forward to the end of the walk through, click on the FAST FORWARD button.
- ⟳ To play the walk through over and over, click on the LOOP button.

If you are finished, save your work as a walk through template. Select the SAVE WALK TEMPLATE command from the WALK menu. Enter a name for the walk through in the dialog box and then click OK. The walk through template will be saved as a part of the drawing file.

Playing a Walk Through in DesignCAD

To play a walk through, display the drawing for which you defined a walk through template. Choose ANIMATION|WALK THROUGH MODE, then WALK|WALK TEMPLATE. Select the appropriate template file, and click OK. You can now click the play button to run your animation.

Hint: DesignCAD remains in Walk Through Mode until you deselect WALK THROUGH MODE under the WALK menu.

See Also: *Animation Mode Command, AVI Command, VRML Command*

Wall Command

Menu: SOLIDS
 Menu Command: WALL
 Shortcut Key: [
 Toolbox Icon: 
 Point 1: First corner of wall
 Point 2: Opposite corner of wall

The Wall command draws a vertical wall of a specific thickness. It is useful when placing walls in a building model.

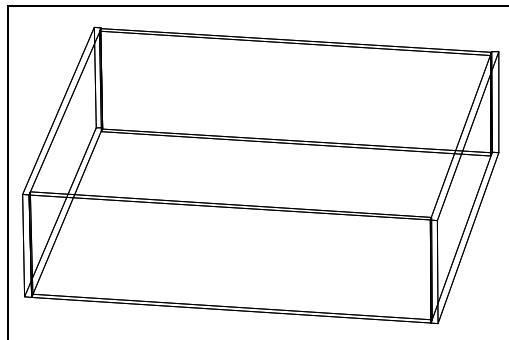
Using the Command

Choose the WALL command. In the WALL THICKNESS box specify the thickness of the wall. Set a point for the first corner of the wall. As you move the cursor, a rubber-band box shows how the wall will be drawn. Set a second point for the opposite corner of the wall face. With the Wall command all you need do is draw a 2-d box, the thickness you enter in the Wall Thickness box is automatically added to the third direction.



Example: Draw a wall that is six inches thick.

For this example, assume that one Drawing Unit equals one foot. Select the WALL command. Enter .5 in the WALL THICKNESS box. Set a point at the first corner. Move the cursor along the XY axis. A rubber-band wall appears, with the cursor location as Point 2. When the wall is the desired size, set a second point. Once the first wall is set, adjacent walls are easily added.



Weld Command

Menu:	EDIT
Submenu:	SELECTION EDIT
Menu Command:	WELD

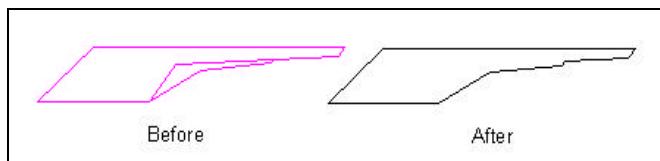
The Weld command combines the area of two or more closed objects that are not solids. These objects must touch or overlap. The combined area may then be manipulated as if it had been drawn that way.

Using the Command

Select the entities to be welded together. Choose the WELD command from the SELECTION EDIT submenu in the EDIT menu. The area of the objects is combined, and any lines that separated the objects are deleted.

Example: Weld two shapes together.

Make sure that the closed entities to be welded together touch or overlap each other and are not solids. Select the objects. Choose the WELD command from the SELECTION EDIT submenu of the EDIT menu. The area of the entities is combined, and any lines that separated the two objects are automatically deleted.



Window Command

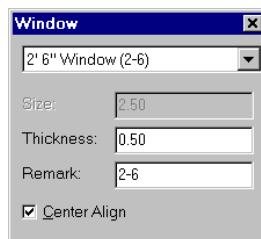
Menu: DRAW
 Menu Command: WINDOW

Point 1: location of the center of the window or one end of the window
 Point 2: placement of dimension text and orientation of the window, if necessary

The Window command inserts a standard architectural symbol for a window.

Using the Command

Choose the WINDOW command from the DRAW menu. The Window dialog box appears.



Select the desired window size or Custom from the list box. If you have chosen the Custom option, enter the size in the SIZE box. By default, the window size appears in the REMARK box unless Custom is selected. However, any information can be entered in the Remark box to be inserted into the drawing with the window. The Thickness is 0.50 by default. Change the Thickness if desired. Check the CENTER ALIGN box to align the window by the center or uncheck it to align the window by one end.

Set the first point for the window. A rubber-band window appears. If the Center Align option is checked, the second point will only determine the location of the dimension text. If the Center Align option is unchecked, the second point will determine the orientation of the window and the location of the dimension text. When the window is positioned to your liking, set the second point. The Line or Double Line into which the window is placed will automatically be cut and capped.

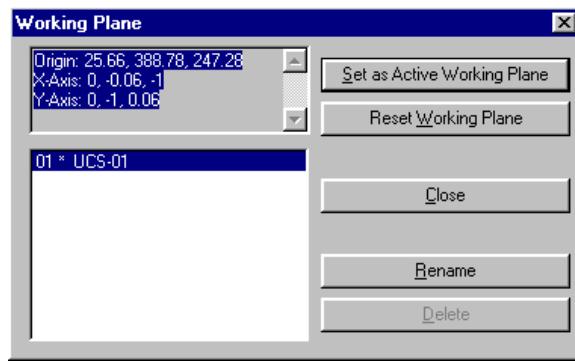
Working Plane Properties Command

Menu: VIEW
 Submenu: WORKING PLANE
 Menu Command: WORKING PLANE PROPERTIES

The Working Plane Properties command makes it easy to switch from one workplane to another when multiple workplanes have been defined for a drawing.

Using the Command

After at least one workplane has been defined (in addition to the default workplane) with the SET WORKING PLANE command, select the WORKING PLANE PROPERTIES command. The Working Plane dialog box appears.



The lower window on the left of the dialog box shows a list of the defined workplanes. The top window on the left gives the specifics for the selected workplane.

Set as Active Working Plane

Clicking this button sets the selected workplane as the current workplane.

Reset Working Plane

Clicking this button resets the default workplane as the current workplane.

Close

Clicking this button closes the dialog box.

Rename

To rename the currently selected workplane, click the RENAME button. A dialog box will appear allowing you to rename the workplane and/or change its description.

Delete

Clicking this button deletes the selected workplane.

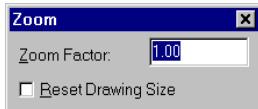
Zoom Command

Menu:	VIEW
Menu Command:	ZOOM
Toolbox Icon:	
Point 1:	Center of zoom

The Zoom command makes the drawing appear larger or smaller on the screen. It does not affect the actual size or scale of the objects in the drawing unless you choose the RESET DRAWING SIZE option.

Using the Command

Choose the Zoom command. In the dialog box set the ZOOM FACTOR and if you want to change the actual drawing size, choose the RESET DRAWING SIZE box.

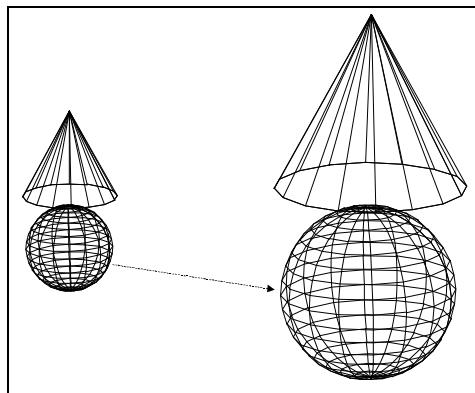


The zoom factor is relative to the current size of the drawing. If you zoom with a factor of two, the drawing appears twice as large. If you zoom with a factor of 0.25, the drawing is displayed at one fourth its current size.

If the RESET DRAWING SIZE box is checked, DesignCAD resizes the objects in the drawing according to the zoom factor. Otherwise, only the apparent size of the objects are changed.

Example: Make your drawing two times larger.

Select the ZOOM command and enter **2** in the ZOOM FACTOR box. Set a point for the center of the zoom. The objects are redrawn, doubled in size.



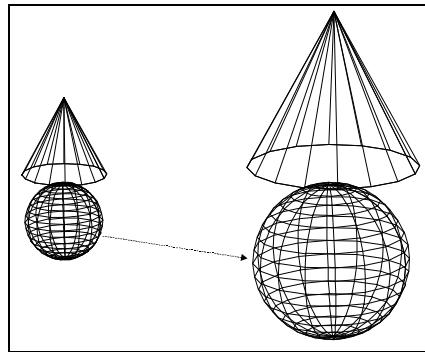
Zoom In Command

Menu:	VIEW
Menu Command:	ZOOM IN
Shortcut Key:	+
Toolbox Icon:	
Point 1:	Center of zoom

The Zoom In command gives you a quick way to zoom into your drawing.

Using the Command

Select the ZOOM IN command. Then position the cursor at the zoom center and click the left mouse button.



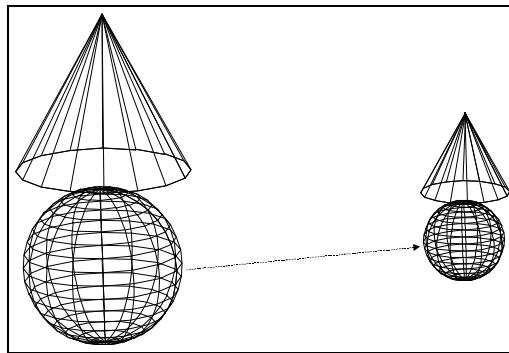
Zoom Out Command

Menu:	VIEW
Menu Command:	ZOOM OUT
Shortcut Key:	-
Toolbox Icon:	
Point 1:	Center of zoom

The Zoom Out command gives you a quick way to zoom out, or reduce the size of the drawing as it appears on the screen.

Using the Command

Select the ZOOM OUT command and position the cursor at the zoom center. Then click the left mouse button.



Zoom Previous Command

Menu:	VIEW
Menu Command:	ZOOM PREVIOUS
Shortcut Key:	Ctrl+M
Toolbox Icon:	

The Zoom Previous command is used to revert instantly back to the zoom setting you used last. If you select this command repeatedly, it goes backwards through the entire sequence of zoom operations since the last time you saved the drawing.

Using the Command

Choose the ZOOM PREVIOUS command. The view returns to the previous zoom factor.

See Also: **Zoom Command, Zoom In Command, Zoom Out Command, Zoom Window Command**

Zoom Redo Command

Menu:	VIEW
Menu Command:	ZOOM REDO
Shortcut Key:	Ctrl+Shift+M
Toolbox Icon:	

The Zoom Redo command is used to revert to the last zoom setting before the Zoom Previous command. For example, if you change your mind about a zoom setting after using the Zoom Previous command, you can select the Zoom Redo command to cancel that zoom action.

Using the Command

Choose the ZOOM REDO command. The view reverts to the view before the Zoom Previous command was used.

Hint: The Zoom Redo command is not available unless you have used the Zoom Previous command.

See Also: **Zoom Previous Command**

Zoom to Selection Command

Menu:	VIEW
Menu Command:	ZOOM TO SELECTION

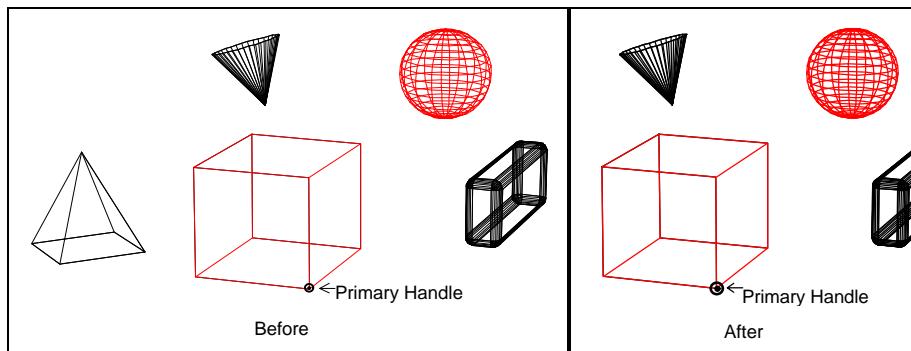
The Zoom to Selection command fits the currently selected entity or entities in a drawing to all view windows. The drawing is displayed at as large a zoom factor as possible while maintaining the visibility of all of the selected entities. If only one entity is selected, it will be centered in all the view windows. Objects or portions of objects that are not selected may be visible, depending on their proximity to the selected entities.

Using the Command

Select an entity in the drawing. Choose the ZOOM TO SELECTION command. The drawing is zoomed so that the selected entity is displayed as large as possible while keeping the entire selected entity visible and centered in all the views.

Example: Zoom in on a box and a sphere.

Draw several entities, including a box and a sphere. Select the box. Pressing the **Shift** key, select the sphere too. Choose the ZOOM TO SELECTION command in the VIEW menu. The drawing is displayed at the maximum zoom factor possible while maintaining visibility of the entire box and the entire sphere.



Zoom Window Command

Menu: **VIEW**

Menu Command: **ZOOM WINDOW**

Shortcut Key: **Z**

Toolbox Icon:

Point 1: One corner of the area to be magnified

Point 2: Opposite corner of the area to be magnified

The Zoom Window command zooms in on your drawing, filling the screen with a specified area of a drawing.

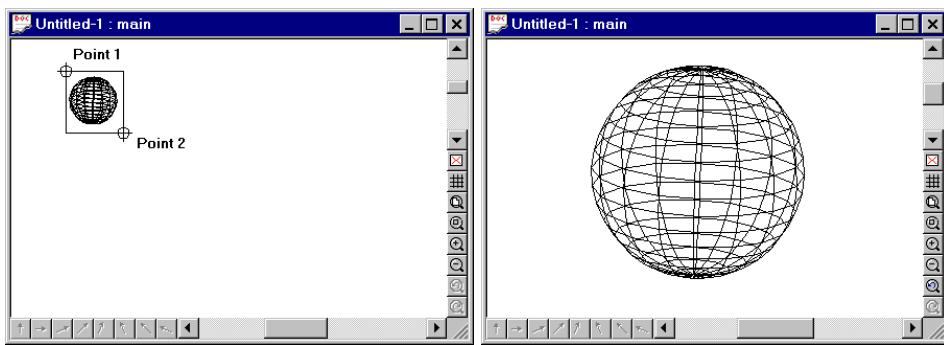
Using the Command

Select the ZOOM WINDOW command. Drag a rectangle around the area you want to zoom into. That rectangle is then enlarged to fill the screen.

The Zoom Previous command can be used to zoom back to the previous size after you use the Zoom Window command.

Example: Zoom in on an area of the screen.

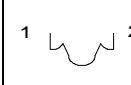
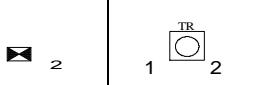
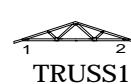
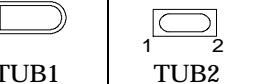
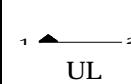
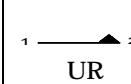
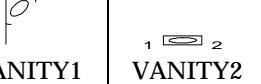
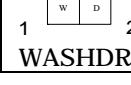
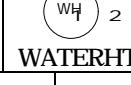
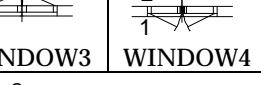
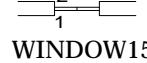
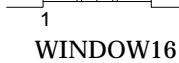
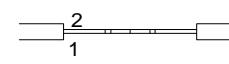
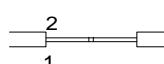
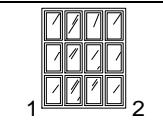
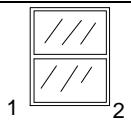
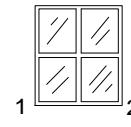
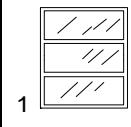
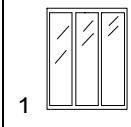
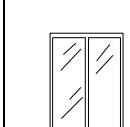
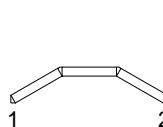
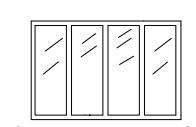
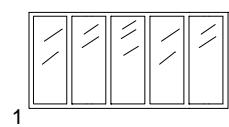
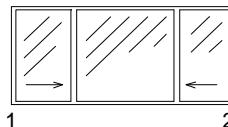
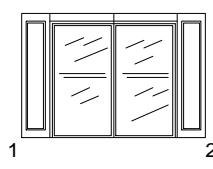
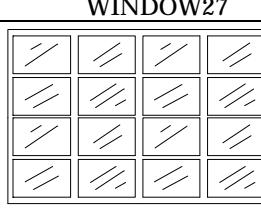
Select the ZOOM WINDOW command. Set a point for one corner of the bounding box. As you move the cursor, a rubber-band bounding box is drawn using the cursor location as Point 2. When you set the second point, the area in the box is redrawn to fit the view window.



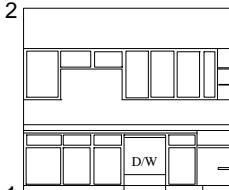
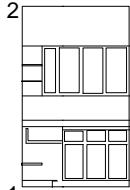
Appendix A: Symbol Libraries

The following symbols are provided with DesignCAD:

ARCHITECTURE Symbol Library

 TOILET3	 TOILET4	 TOILET5	 TOILET6	 TOILET7	 TRANSF1	 TRASH1
 TRUSS1	 TRUSS2	 TRUSS3	 TRUSS4	 TRUSS5	 TUB1	 TUB2
 TUB3	 UL	 UR	 VALVE1	 VALVE2	 VANITY1	 VANITY2
 VANITY3	 WASHDRY	 WATERHTR	 WINDOW1	 WINDOW2	 WINDOW3	 WINDOW4
 WINDOW5		 WINDOW15		 WINDOW16		
 WINDOW17		 WINDOW18		 WINDOW19	 WINDOW20	
 WINDOW2 1	 WINDOW2 2	 WINDOW2 3	 WINDOW24	 WINDOW25	 WINDOW26	
 WINDOW27		 WINDOW28		 WINDOW29		
 WINDOW30		 WOMAN1				

CABINET Symbol Library

						
						
CAB3		CAB4	CAB5			
						
CAB6	CAB7		CAB8	CAB9		
						
CAB10A	CAB10B	CAB10C	CAB10D	CAB10E	CAB11A	CAB11B
						
CAB12A	CAB12B	CAB12C	CAB12D	CAB12E	CAB12F	CAB12G
						
CAB12H	CAB12I	CAB13A	CAB13B	CAB13C	CAB13D	CAB13E
				DRAWER1A	DRAWER1B	OVEN1A
CAB13F	CAB13G	CAB13H	CAB13I			
OVEN1B	RANGE1	SINK1A	SINK1B	SINK1C	SINK1D	SINK2A

SINK2B	SINK2C	UTIL1A	UTIL1B	VAN1A	VAN1B	VAN1C
VAN1D	VAN2A	VAN2B				

ELECTRICAL Symbol Library

BATTERY	BELL	BUZZER	D1	D2	FUSE	GND
HORN	LAMP	LAMP1	LAMP2	LAMP3	LAMP4	LAMP5
LAMP6	LAMP7	LAMP8	LAMP9	LAMP10	LAMP11	LAMP12
LAMP13	LAMP14	LAMP15	LAMP16	LAMP17	LAMP18	MOTOR
OUT1	OUT2	OUT3	OUT4	OUT5	OUT6	OUT7
OUT8	OUT9	OUT10	S1	S2	S3	S4
S5	XFRMR					

ELECTRON Symbol Library

AMP	AND	ANT1	BAT1	BAT2	BRK1	C1
C2	C3	CGND	D1A	D2A	D3	DS1
DS5	F1	GND	IC6	IC8	IC10	IC12
IC14	IC16	IC18	IC20	IC40	JACK	L1
L2	L3	NAND	NOR	NOT	OR	OSC
PLUG	Q1	Q2	Q3	Q4	Q5	Q6
Q7	R1	R2	S1A	S2A	S3A	SPKR
T1	T2	XOR	XTL			

HYD Symbol Library

HYD1	HYD2	HYD3	HYD4	HYD5	HYD6	HYD7
HYD8	HYD9	HYD10	HYD11	HYD12	HYD13	HYD14
HYD15	HYD16	HYD17	HYD18	HYD19	HYD20	HYD21
HYD22	HYD23	HYD24	HYD25	HYD26	HYD27	HYD28
HYD29	HYD30	HYD31	HYD32	HYD33	HYD34	HYD35
HYD36	HYD37	HYD38	HYD39	HYD40	HYD41	HYD42
HYD43	HYD44	HYD45	HYD46	HYD47	HYD48	HYD49
HYD50	HYD51	HYD52	HYD53	HYD54	HYD55	HYD56
HYD57	HYD58	HYD59	HYD60	HYD61	HYD62	HYD63
HYD64	HYD65	HYD66	HYD67	HYD68	HYD69	HYD70
HYD71	HYD72	HYD73	HYD74	HYD75	HYD76	

PIPING Symbol Library

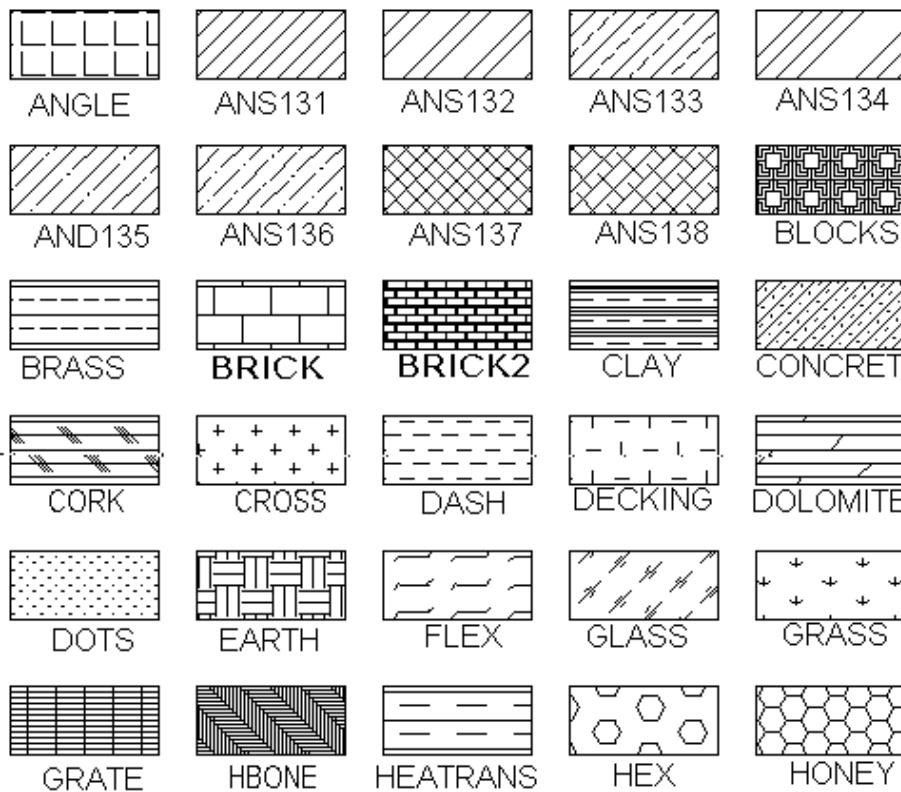
AUTOAIR	BFPREV	CBBRANC H	CREDUCE	CSBRANC H	CTBRANC H	ENDCAP
EREDUCE	EXPJOINT	FIREHYD	FLEX	GASCOCK	HAMMER	HBIBB
HBIBBR	STRBASK	STRWYE	SWFLOW	SWPRESS	TPLUG	UNION
V3WAY	VANGLE	VBALL	VBUTTER	VCHECK	VDIA	VGATE
VGLOBE	VHOSE	VNSCHEC K	VPREDUCE	VPRELIEF	VRISER	VSCHECK
VSOL	VYARD	YARDCLR				

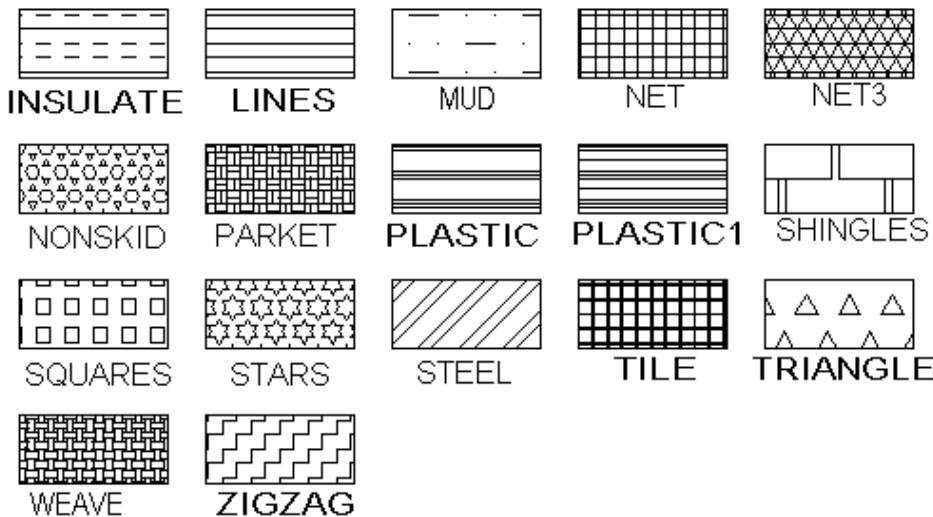
Appendix B: Hatch Patterns

With DesignCAD you can either use the hatch patterns provided with the program or define your own.

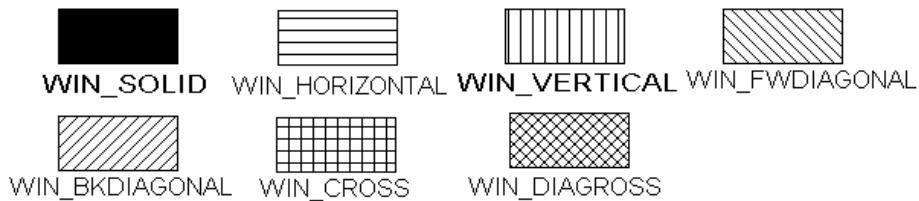
The following hatch patterns are provided with the software and can be selected with the Hatch command:

Scaleable Hatch Patterns





Non-Scaleable Hatch Patterns



Defining Your Own Hatch Pattern

Some of the hatch patterns used by the Hatch, Hatch Line, and Hatch Fill commands are found in the file DCHATCHC.SYS. You can use your own hatch patterns with DesignCAD by adding them to this file. The file DCHATCHC.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCHATCHC.SYS before you modify it.

This file does not contain definitions for the "non-scaleable" hatch patterns such as WIN_SOLID. These are defined by MS Windows.

The file DCHATCHC.SYS consists of a series of hatch patterns. Each hatch pattern is in the following format:

Hatch Pattern Name	Number of Line Definitions	Pattern Scale for Preview
	A B C D E F1 F2 F3 F4 F5 F6	
(there will be a line of these values for each line definition)		

The *Number of Line Definitions* indicates the number of separate line segments that make up the pattern. The *Pattern Scale for Preview* is the pattern scale that particular hatch pattern uses in the Preview box.

Line Definitions values:

- A. Angle (0=horizontal, 90=vertical)
- B. X offset of first occurrence (*relative to an arbitrary starting point*)
- C. Y offset of first occurrence (*relative to an arbitrary starting point*)
- D. X change from first to second occurrence. (*relative to B,C*)
- E. Y change from first to second occurrence. (*relative to B,C*)
- F. six numbers defining the line pattern:
 - 1) length on
 - 2) length off
 - 3) length on
 - 4) length off
 - 5) length on
 - 6) length off

"X" and "Y" are relative to the angle of the line. This means that if the angle is 90, then X is actually the relative vertical displacement, and Y is the relative horizontal displacement.

For example, the hatch pattern definition for the BRASS pattern is:

```
BRASS
2 45
0 0 0 0 20 40 0 0 0 0 0
0 0 10 0 20 10 5 0 0 0 0
```

There are two lines in the pattern. The first line definition is oriented at an angle of zero, so it is a horizontal line. It is a solid line, since there is only one non-zero value in the last six numbers. This first line starts at 0, 0 (an arbitrary position), and it will repeat 20 "units" above this line. This line ends 40 "units" from this starting point.

The next line is also a horizontal line, but it starts 10 units above the first line. It is a dashed line, repeating a pattern of 10 "on" and 5 "off."

A more complicated example is the BRICK2 pattern:

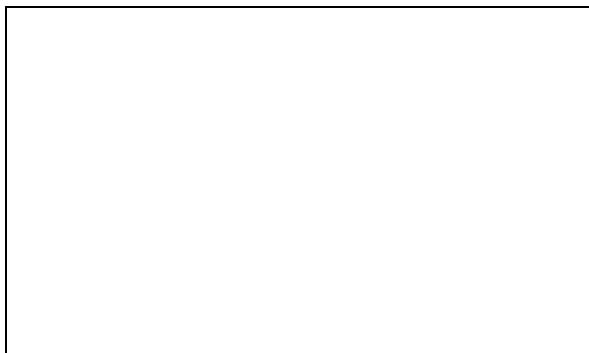
```
BRICK2
4 20
0 0 0 7 6 13 1 0 0 0 0
0 0 5 7 6 13 1 0 0 0 0
90 0 0 6 7 5 7 0 0 0 0
90 0 1 6 7 5 7 0 0 0 0
```

This pattern consists of 4 lines, 2 horizontal, and 2 vertical.

The first line is a horizontal, dashed line, repeating 13 on, and 1 off. It starts at 0,0, and will repeat at 7 units over and 6 units up. The second line is the same as the first, except that it starts 5 units above the first line.

The third line is a vertical line, starting at 0,0, repeating with 5 on and 7 off. Note that since it is vertical, the line actually repeats at 7 over and 6 up, not 6 over and 7 up. Because the fourth line is at a 90 degree angle, it starts at 1 unit to the left of the starting point.

If viewed by themselves, the four line segments will look something like this:



Appendix C: Creating Custom Line Styles and Shapes

Creating a Custom Line Style

Some of the line styles used by DesignCAD are found in the DCLSTYLE.SYS file located in the DesignCAD directory. You can create your own custom line styles for use in DesignCAD by adding them to this file. The file DCLSTYLE.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCLSTYLE.SYS before you modify it.

The file DCLSTYLE.SYS consists of a series of line styles. Each line style is in the following format:

Line Style Name (*no more than 30 characters long*); Text Comment
(*optional*)
Number of Lines and Shapes
Line or shape definition (*there will be a separate line of values for each line or shape*)

The *Number of Lines and Shapes* indicates the number of separate line segments and shapes that make up the line style.

Line Definition values:

- A. *PD, PU*
 - 1) *PD* = length of pen down (*line segment*)
 - 2) *PU* = length of pen up (*space after line segment*)

Shape Definition values:

- A. [S(*width, height*) R(*z angle*) P(*x,y*) ; %%^%-% | *Shape Name*]
 - 1) *S* = size of shape; original size is used if size is not specified; if only one value is specified, *width* and *height* are the same. (*optional*)
 - 2) *R* = rotation; original angle is used if rotation is not specified. (*optional*)
 - 3) *P* = position of shape relative to the end of the previous Line or Shape definition; if only one value is specified, it is assigned to *x*. (*optional*)

Note: When these commands are present, Scale should be used first, then Rotation, and finally Position.

- 4) %^ = shape orientation won't be affected by the order of points of the line. (*optional*)
- 5) %- = ignores the bold flag of the line. (*optional*)
- 6) %| = shape does not rotate with the line segment.

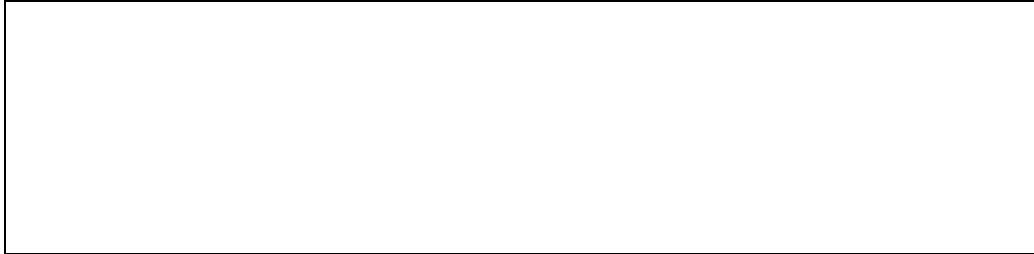
Note: If more than one of %^, %-, and %| are used, there should not be a space between them.

7) *Shape Name* = shape to be used in line style; must be defined in DCSHAPE.SYS.

The following is an example of a simple line style that has only one line segment definition.

```
Long Dash
1
50, 10
```

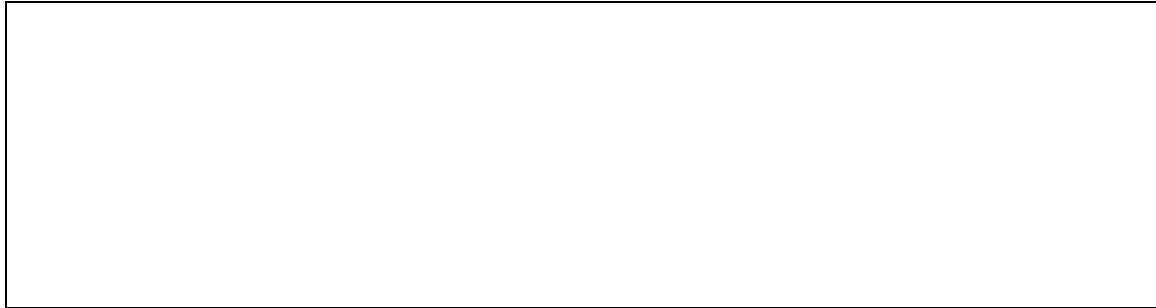
This simple line style will look like this:



The next example is a little more complicated. Notice that the only thing that changed in the first three lines between this example and the last one are the Line Style Name and the Number of Lines and Shapes has changed from "1" to "2." The same long dash (50) and small space (10) are still in the line style.

```
Long Dash and Star
2
50, 10
[P(-35,0) ; Star], 0
```

The additional line of the "Long Dash and Star" line style inserts a "Star" shape (defined in the DCSHAPE.SYS file). All of the information for the "Star" shape is entered as the fourth line of the "Long Dash and Star" line style [P(-35,0) ; Star], 0. The center of the star is placed so that its center is centered on the line segment defined in the third line of the "Long Dash and Star" line style (-35 = 50/2+10, the value 50 = length of the original dash; the value 10 = amount of space that follows the dash in the line segment definition). This line style will look like this:



Now a more complicated line style from scratch.

```
DCAD
2
2, 1
[ ; DCAD], 20
```

Doesn't look that complicated does it? It's really not. It just requires that you add a shape to the DCSHAPE.SYS file.

Note: Be sure that the last line of both the DCLSTYLE.SYS and DCSHAPE.SYS is an empty carriage return.

Creating a Custom Line Style Shape

The shapes that can be used by DesignCAD line styles are found in the DCSHAPE.SYS file located in the DesignCAD directory. You can create your own custom shapes for use in DesignCAD line styles by adding them to this file. The file DCSHAPE.SYS is an ASCII file that can be edited with a text editor or a word processor in ASCII mode. Be sure you have a second copy of the file DCSHAPE.SYS before you modify it.

Each shape definition is in the following format:

```
Shape Name (no more than 30 characters long); Text Comment (optional)
Command and its number of points
Points for command (points appear on separate lines and consist of x and y
coordinates; x and y coordinates may be separated by a space or a comma)
Additional commands and points
* (indicates the end of the shape)
```

Commands allowed in the DCSHAPE.SYS file are:

- 1) A = Three-point arc with center, starting, and ending points

Note: Identical starting and ending points make a circle.

- 2) C = Spline curve
- 3) V = Polyline

The best way to make a new shape is to record a macro and use the Arc (Center, Begin, end), Curve, and Line commands to draw the shape. Once the drawing of the shape has been recorded as a macro, you can open the macro file, delete the information that is not needed for the DCSHAPE.SYS file, and change the command names to the DCSHAPE.SYS format.

Here is a portion of the recorded macro and the equivalent portion of what was added to the DCSHAPE.SYS file to create the DCAD shape (*in this comparison, blank lines in the DCAD shape definition are used only to keep the code comparison parallel; there should **not** be blank lines in the actual DCSHAPE.SYS file*):

```
>SetStartPoint                               DCAD
{
}
```

```
>Line                                         v 2
{
  <PointXYZ 0.0000,0.0000,0.0000          0.0000,0.0000
  <PointXYZ 0.0000,4.0000,0.0000          0.0000,4.0000
}

>Curve                                         c 3
{
  <PointXYZ 0.0000,4.0000,0.0000          0.0000,4.0000
  <PointXYZ 1.7500,3.8000,0.0000          1.7500,3.8000
  <PointXYZ 2.7500,3.0000,0.0000          2.7500,3.0000
  <Type 0
}

>Line                                         v 2
{
  <PointXYZ 2.7500,3.0000,0.0000          2.7500,3.0000
  <PointXYZ 2.7500,1.0000,0.0000          2.7500,1.0000
}

>Curve                                         c 3
{
  <PointXYZ 0.0000,0.0000,0.0000          0.0000,0.0000
  <PointXYZ 1.7500,0.2000,0.0000          1.7500,0.2000
  <PointXYZ 2.7500,1.0000,0.0000          2.7500,1.0000
  <Type 0
}
...
...
```

Don't forget to end the shape definition with an asterisk (*).

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